Geomage g-Space™ User Manual

USER MANUAL

© 2026 Geomage 2003 LTD
Geomage 2003 LTD

1 Introduction 9

2 Installation 13

2.1 Requirements ... 14
2.2 Installation steps ... 14
2.3 Licensing ... 23
2.4 Software update ... 27

3 User Interface 29

3.1 Ribbon bar ... 33
3.1.1 Main bar ... 35
3.1.2 Views bar ... 38
3.1.3 Database ... 41
3.1.4 Seismic bar ... 44
3.1.5 Wells bar ... 46
3.1.6 Petrophysics ... 49
3.1.6.1 Base properties ... 55
3.1.6.2 Shale Volume ... 58
3.1.6.3 Porosity ... 65
3.1.6.4 Saturation ... 69
3.1.6.5 Pore pressure ... 73
3.1.6.6 Lithology ... 77
3.1.6.7 ML Log Prediction ... 82
3.1.6.8 Mineral Volume Estimation ... 88
3.1.7 Well tie bar ... 93
3.1.8 Interpretation ... 96
3.1.9 Velocity model bar ... 99
3.1.10 Modelling ... 100
3.1.11 Attributes bar ... 105
3.1.12 Statistical analysis bar ... 111
3.1.13 Mis tie seismic bar ... 117
3.1.14 AVO bar ... 119
3.1.15 Reservoir bar ... 122
3.2 Panels ... 123
3.2.1 Data Manager ... 124
3.2.1.1 Seismic 2D ... 132
3.2.1.2 Seismic 3D ... 134
3.2.1.3 Arbitrary lines ... 135
3.2.1.4 Wells ... 137
3.2.1.4.1 Create curve ... 140
3.2.1.4.2 Load raster ... 142

© 2026 Geomage 2003 LTD

3.2.1.4.3 Scale Curves ... 143
3.2.1.4.4 Extract seismic trace ... 147
3.2.1.5 Well sections ... 150
3.2.1.6 Markers ... 150
3.2.1.7 Stratigraphy ... 158
3.2.1.8 Horizons ... 158
3.2.1.9 Faults ... 158
3.2.1.10 Topography ... 160
3.2.1.11 Raster files ... 161
3.2.1.11.1 Satellite imagery downloader ... 161
3.2.1.12 Shapefiles layers ... 162
3.2.1.13 Maps ... 162
3.2.1.14 Structural Models ... 163
3.2.1.15 Velocity model ... 167
3.2.1.16 Polygons ... 167
3.2.1.17 3D Grids ... 169
3.2.1.17.1 3D Grid Quality Control ... 172
3.2.1.17.2 Grid calculator ... 175
3.2.1.18 Geo-bodies ... 177
3.2.1.19 Multi-Z Surfaces ... 178
3.2.1.20 Wavelets ... 180
3.2.1.21 Fault model ... 181
3.2.1.22 Points ... 185
3.2.2 Filter wizard ... 188
3.2.3 View Manager ... 189
3.2.4 Conceptual Model ... 191
3.2.4.1 Fluid contacts ... 195
3.2.5 Module Properties ... 197
3.2.6 Visual Settings ... 206
3.2.6.1 View ... 208
3.2.6.2 Object ... 210
3.2.7 Actions ... 214
3.2.8 Output View ... 219
3.2.9 Progress View ... 221
3.2.10 Object properties ... 221
3.3 Workspace ... 224
3.3.1 Views synchronization ... 228
3.3.2 Seismic data workspace ... 228
3.3.3 Wells data workspace ... 232
3.4 Views ... 233
3.4.1 Location map ... 235
3.4.2 Seismic view ... 240
3.4.3 3D view ... 244
3.4.4 AVO views ... 272

© 2026 Geomage 2003 LTD

3.4.4.1 Image gather ... 273
3.4.4.2 AVA gather ... 276
3.4.4.3 3D image gathers ... 277
3.4.4.4 AVO cross-plot ... 279
3.4.4.5 AVO graph ... 280
3.4.4.6 Stack gather ... 283
3.4.5 Table view ... 285
3.4.6 Well tie view ... 287
3.4.7 Well correlation view ... 296
3.4.7.1 LAS editing ... 320
3.4.7.2 Working with Palettes ... 323
3.4.8 Graphs ... 326
3.4.8.1 Histogram view ... 327
3.4.8.2 Cross plot view ... 330
3.4.8.3 Pine graph view ... 332
3.4.8.4 Variogram view ... 335
3.4.8.5 Stereonet view ... 336
3.4.8.6 Tornado view ... 338
3.4.9 Map editor ... 341
3.5 Controls and shortcuts ... 343
3.6 Seismic data ... 344

4 Getting Started 355

4.1 Creating a new project ... 357
4.2 Saving the project ... 359
4.3 Loading an existing project ... 361
4.4 Closing the project ... 362
4.5 Project configuration ... 363
4.5.1 Coordinate and measurement system ... 365
4.5.2 Datum configuration ... 368
4.6 Demo project ... 369
4.7 Remote database ... 370

5 Import data 372

5.1 Import Seismic ... 373
5.1.1 Import attributes ... 382
5.1.2 Import ZGY ... 383
5.1.3 Convert ASCII to seismic ... 384
5.1.4 Import mistie cross-points ... 386
5.1.5 Import mute picking ... 387
5.1.6 Import GSD ... 389
5.1.7 Import other seismic formats ... 390

© 2026 Geomage 2003 LTD

5.2 Import Wells 391
5.2.1 Create well 392
5.2.2 Import well headers 393
5.2.3 Import inclinometry 397
5.2.4 Import markers 403
5.2.5 Import markers structure 406
5.2.6 Import LAS files 407
5.2.7 Import checkshots 414
5.2.8 Import comment log 421
5.2.9 Import DLIS 426
5.2.10 Import WITSML 427
5.2.11 Import PRODML 430
5.2.12 Import from OSDU 432
5.2.13 Import stratigraphy 433
5.2.14 Import marker ages 434
5.2.15 Import well log images 436
5.3 Import Horizons 438
5.4 Import Faults 441
5.5 Import Shapefile 443
5.6 Import polygons 444
5.7 Import topography 445
5.8 Import images 446
5.8.1 Import satellite imagery 449
5.9 Import maps 450
5.10 Import grids 452
5.10.1 Import 3D models 453
5.10.2 Import RESQML 456
5.10.3 Import DXF 458
5.11 Import points 460
5.12 Import from g-Platform 461

6 Export data 463
6.1 Export well headers 464
6.2 Export inclinometry 466
6.3 Export markers 468
6.4 Export markers structure 469
6.5 Export checkshots 471
6.6 Export las 474

© 2026 Geomage 2003 LTD

6.7 Export comment log 477
6.8 Export horizons 478
6.9 Export faults 480
6.10 Export maps 482
6.11 Export grids 484
6.11.1 Export 3D models 485
6.11.2 Export RESQML 486
6.11.3 Export to VTK 488
6.11.4 Export DXF 489
6.12 Export polygons 491
6.13 Export mistie cross-points 491
6.14 Export mute picking 493
6.15 Export seismic 494
6.16 Export seismic to GeoTIFF 496
6.17 Export marker ages 497
6.18 Export well logs to Excel 498
6.19 Export points 499
6.20 Export shapefile 500
6.21 Export view image 502
6.22 Export to g-Platform 503

7 Formats 505

7.1 Well Data Formats 507
7.2 Seismic Data Formats 511
7.3 Horizon Data Formats 513
7.4 Maps Data Formats 514
7.5 Fault Data Formats 515
7.6 Image Data Formats 516
7.7 GIS Data Formats 517
7.8 Grid Data Formats 519
7.9 GOCAD Data Formats 519
7.10 Trace headers math 521
7.11 Model Data Formats 522
7.12 Point Data Formats 523

8 Service Tasks 525

© 2026 Geomage 2003 LTD

9 Best Practice Flows 527

9.1 Workflows ... 535

9.2 Well log correlation ... 538

9.2.1 Setting Up a Well Template ... 544

9.2.2 Working with Markers ... 551

9.2.3 Well log calculator ... 558

9.3 Lithology Interpretation ... 566

9.4 Well-Tie ... 567

9.5 Seismic picking ... 586

9.5.1 Seismic data visualization ... 587

9.5.1.1 2D data ... 589

9.5.1.2 3D data ... 599

9.5.2 Horizon picking ... 614

9.5.3 Fault picking ... 627

9.5.3.1 Fault prediction ... 640

9.5.3.2 AI Faults Wizard ... 642

9.6 Velocity model workflow ... 658

9.6.1 Setting up the velocity model ... 660

9.6.1.1 Constant ... 662

9.6.1.2 Constant layered ... 663

9.6.1.3 Markers + horizons ... 664

9.6.1.4 Grid velocity by markers + horizons ... 666

9.6.1.5 Checkshot extrapolation ... 667

9.6.1.6 Checkshot + markers ... 669

9.6.1.7 Checkshot + markers + horizons ... 670

9.6.1.8 From Vrms ... 672

9.6.2 Velocity model QC ... 675

9.6.3 Import velocity model ... 678

9.6.4 Export velocity model ... 679

9.7 Seismic attributes ... 680

9.7.1 Spectral decomposition ... 688

9.8 Gather conditioning ... 692

9.9 AVO ... 700

9.10 Mis-tie correction ... 713

9.10.1 2D mis-ties correction ... 715

9.10.2 3D mis-ties correction ... 720

9.11 Mapping ... 724

9.11.1 Map creating ... 727

9.11.2 Map building ... 730

9.11.3 Map editing ... 733

© 2026 Geomage 2003 LTD

9.11.4 Residual calculation 737
9.12 Statistical analysis 742
9.12.1 Cross plot 745
9.12.2 Histogram 748
9.12.3 Variogram 752
9.12.4 Stereonet 756
9.13 Geological modeling 758
9.14 3D grid creating 765
9.15 Mesh optimization 773
9.16 Properties modeling 776
9.16.1 Seismic Data Upscaling 782
9.16.2 Upscale 3D Grid 785
9.17 Volumetric calculations 790
9.18 Multi-user mode 795
9.19 Structural modeling 798
9.19.1 Structural Uncertainty Analysis 804
9.20 Python scripting 811
9.21 Well Data QC and Verification 814

Index 816

© 2026 Geomage 2003 LTD

Introduction

1 Introduction

Geomage g-Space™ application designed as a full interpretation software, both geological and geophysical.

Subscribe to our channel and watch g-Space overview on YouTube

Geological Interpretation package
The first version released in November 2021 built of Seismic and Geological interpretation packages that includes all of required features needed for accurate, fast and convenient structural interpretation process.

View-only package:
- Loading 2D and 3D post-stack seismic data, including trace arithmetic
- Loading Well data, such as headers, markers, checkshots, las files, inclinometry
- Loading GIS shapefile and image data
- 2D displays in time domain

Common features:
- Support for Metric or Imperial measurement system
- Import/export Well data, such as headers, markers, checkshots, las files, inclinometry
- Advanced 2D and 3D views in time and depth domains
- Import GIS shapefiles and cultural data
- Import horizons, faults, grids
- CRS conversion
- Import geo-referenced image data
- Stratigraphy chart for markers, horizons and wells
- Map calculator and editor
- DEMO project based on Teapot Dome data

Seismic interpretation package:
- Loading 2D and 3D post-stack seismic data, including trace arithmetic
- Seismic mis-tie workflow
- Well-tie workflow
- Advanced arbitrary line creation and visualization by line or coordinates picking
- Horizon picking/import/export
- Fault picking/import/export, both in 2D or 3D views
- Seismic ghost view
- Flattening to horizon
- Mapping, including fault consideration
- Velocity model building and conversion by
  - checkshots

© 2026 Geomage 2003 LTD

  - checkshots+markers by zones
  - checkshots+markers+horizons by zones
- Pine chart / velocity model QC
- Depth conversion
- Spectral analysis - single and multi-zone

Geological interpretation package:
- Well arbitrary section
- Lithology/discrete log visualization
- Curve ghost view
- Flattening to marker/datum
- Well correlation / Markers picking
- Crossplot data analysis
- Well-log calculator
- Map interpolation by markers
- Mapping, including fault consideration
- Gross volume calculation
- Geobody creation

Reservoir interpretation package:
- 3D Static modeling using different algorithms
- 3D Gridding
- Well log propagation
- Volumetric / GRV calculation

Geophysical interpretation package:
- Import prestack gathers 2D/3D
- Gather conditioning, including trim statics, NMO, RMO and parabolic Radon processing
- Polynomial analysis of AVO
- Intercept and gradient
- Top/bottom and angle-range muting of gathers
- Stacking of gathers, including common-offset and angle stacks
- Spectral balancing and Q compensation
- Predictive and wavelet-based deconvolution
- FXY decon filtering and despiking
- Concatenating gathers and seismic files
- Spectral decomposition
- Instantaneous attributes
- Band-pass filtering, AGC and gain (stacks and gathers)
- Advanced 3D prestack gathers view

Multi-user package:
- Multi-user management

© 2026 Geomage 2003 LTD

- Multi-user interpretation QC tools
- Centralized project management
- PostgreSQL database
- Revision history
- Monitored user access

© 2026 Geomage 2003 LTD

Installation

2 Installation

This paragraph gives the general information about:

System requirements 14 - the minimum requirement of the system for comfortable work
Installation steps 14 - guides through the installation process
Licensing 23 - guides through the licensing process and different license options
Software update 27 - explains how to get the latest g-Space version.

To download the latest version of g-Space please follow the link:
https://www.geomage.com/seismic-software/

2.1 Requirements

General assumptions for workstation are:

Operating system – 64-bit Only

- Windows: 7/8/10
- Linux and MacOS - please contact Geomage for updated status of release for those OS.

CPU

- 1 core minimum. Since g-Space designed to work in multi-thread mode in order to get better performance, it is advised to have as many CPU cores as possible to achieve best performance during calculation processes.

Disk

- SSD storage for seismic prestack data (recommended)
- HDD drive for the rest of the project data

RAM

- 4GB minimum
- the best practice is to have 2 times of loaded seismic data size

2.2 Installation steps

Installation steps for g-Space

Windows version

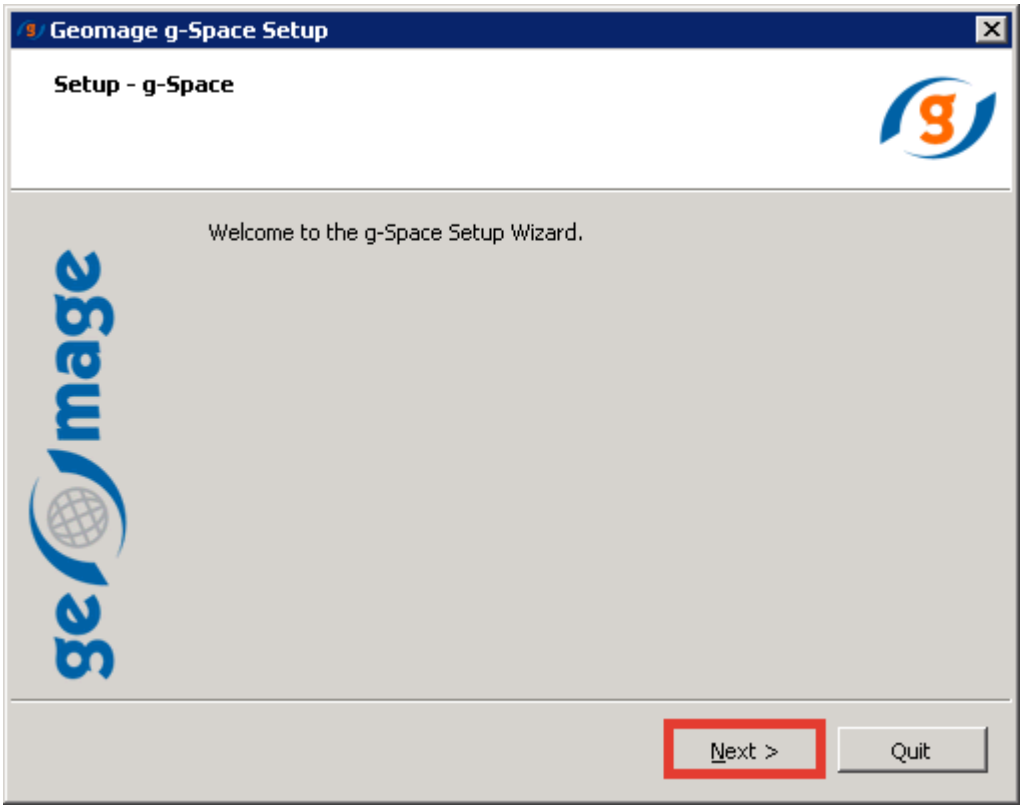
Geomage g-Space can be installed either from the company's web-site or directly from offline installation file.

© 2026 Geomage 2003 LTD

If you have issues with internet access, please contact Geomage support at support@geomage.com and get instruction of how to get offline installation file.

Step 1

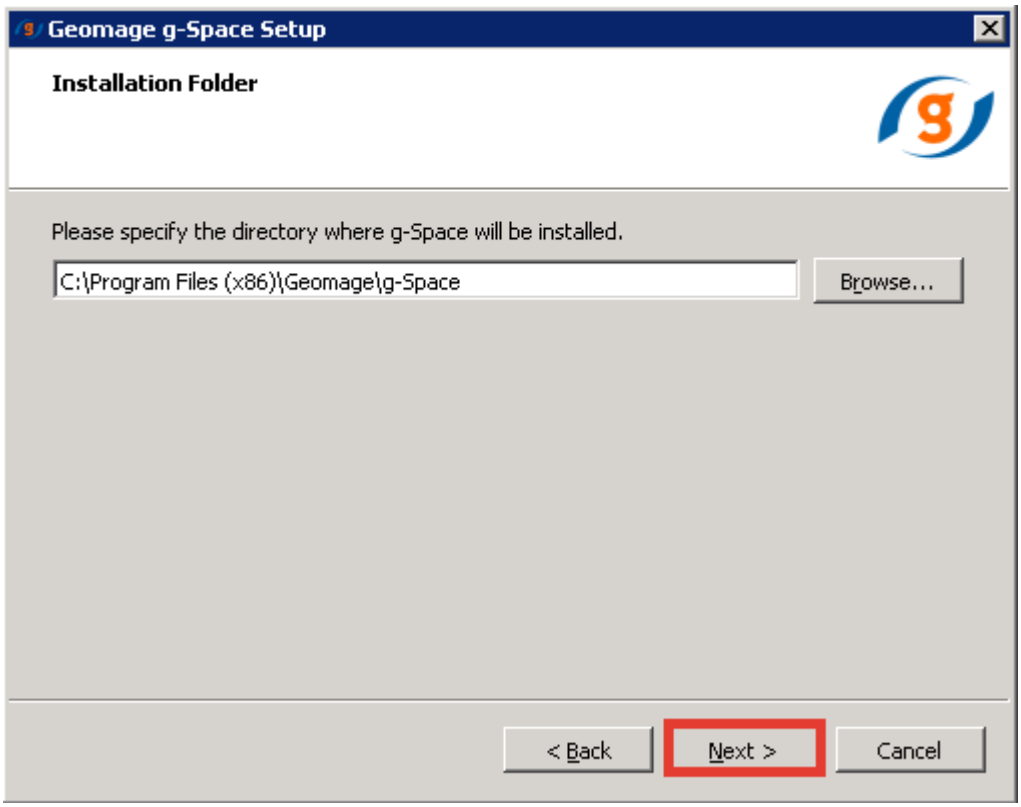
Make sure you're **logged in as an administrator** of the workstation and run the downloaded EXE file [GSpaceInstaller_Online_Win_xxxxx.exe] to start the installation. Click **Next**.



Step 2

Specify the installation directory for g-Space application and click **Next**.

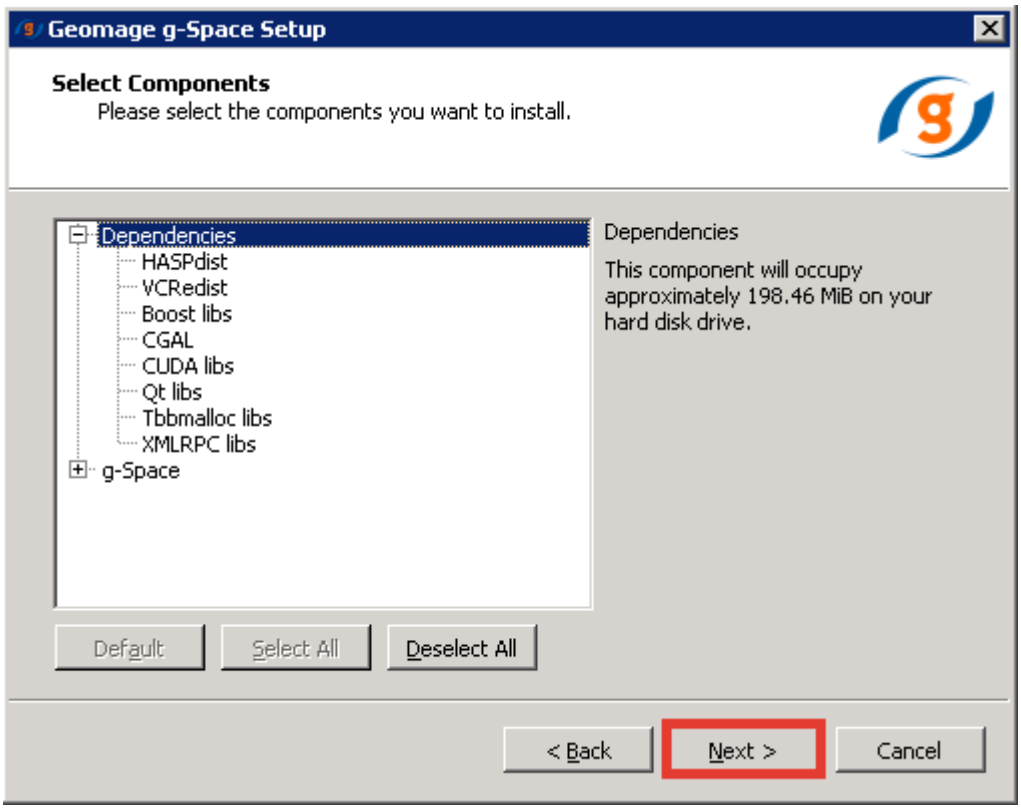
© 2026 Geomage 2003 LTD



Step 3

Select the requested components for installation (by default all options are checked) and click "Next".

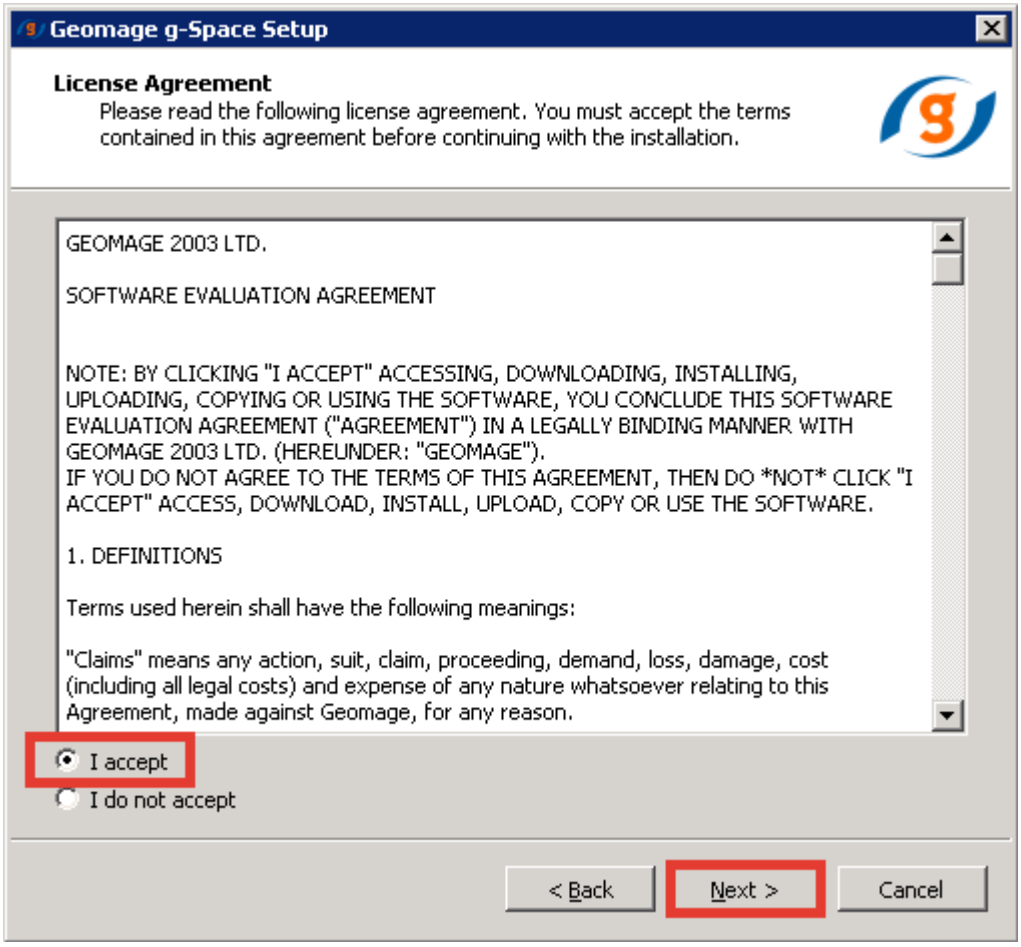
© 2026 Geomage 2003 LTD



Step 4

Review the "License Agreement", check the box to accept and click Next.

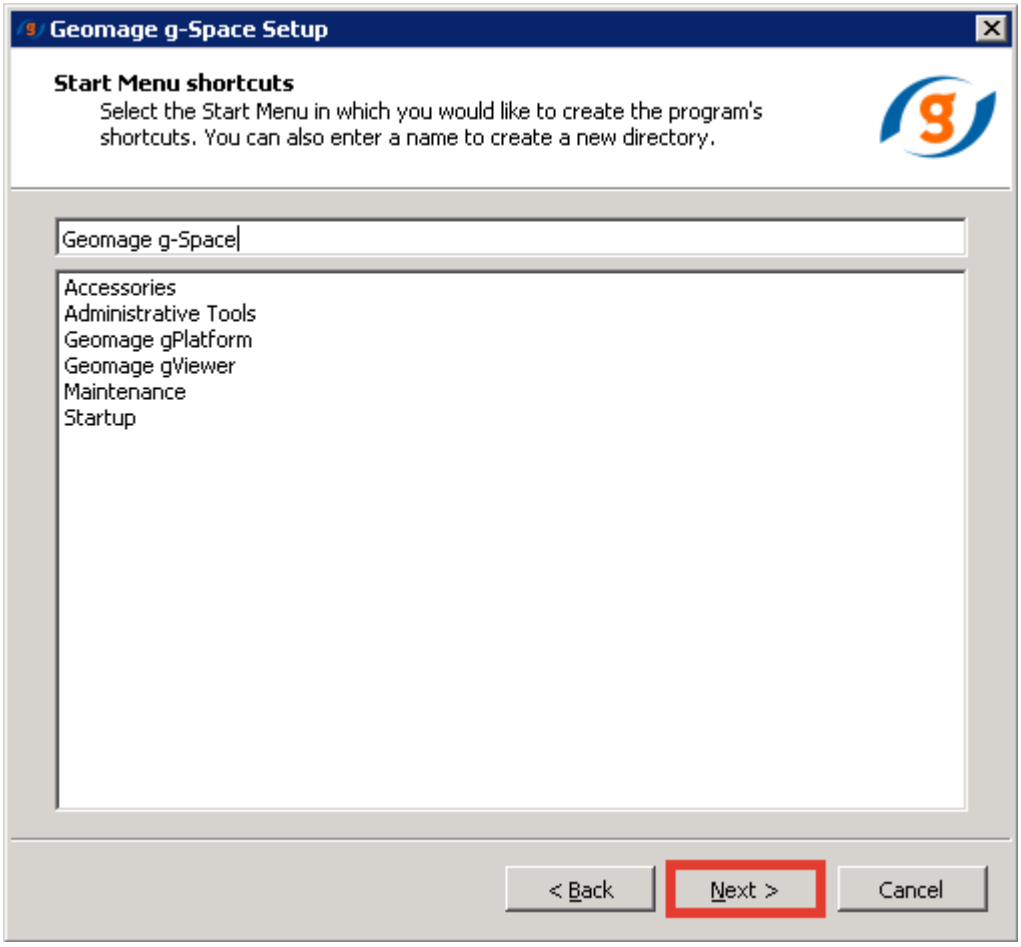
© 2026 Geomage 2003 LTD



Step 5

Select the "Start Menu" for program's shortcuts and click **Next**.

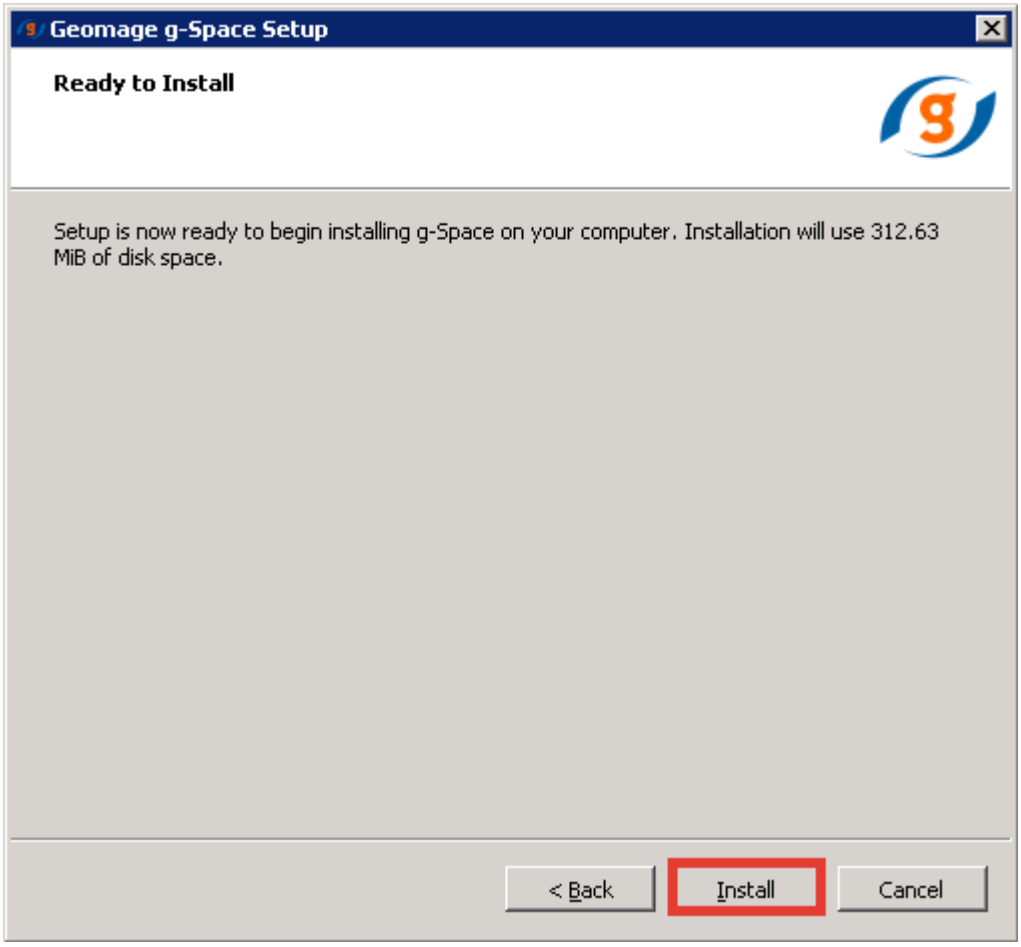
© 2026 Geomage 2003 LTD



Step 6

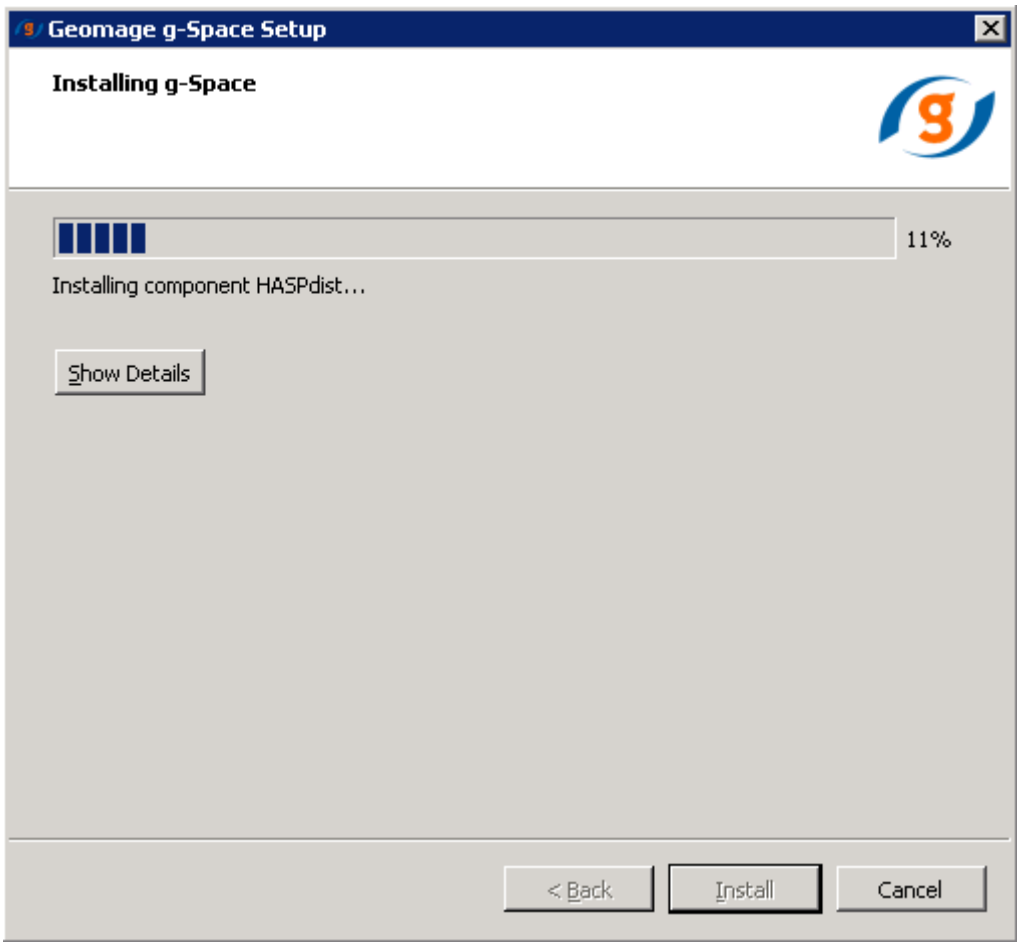
g-Space is ready to be installed, click **Install** to begin the installation.

© 2026 Geomage 2003 LTD



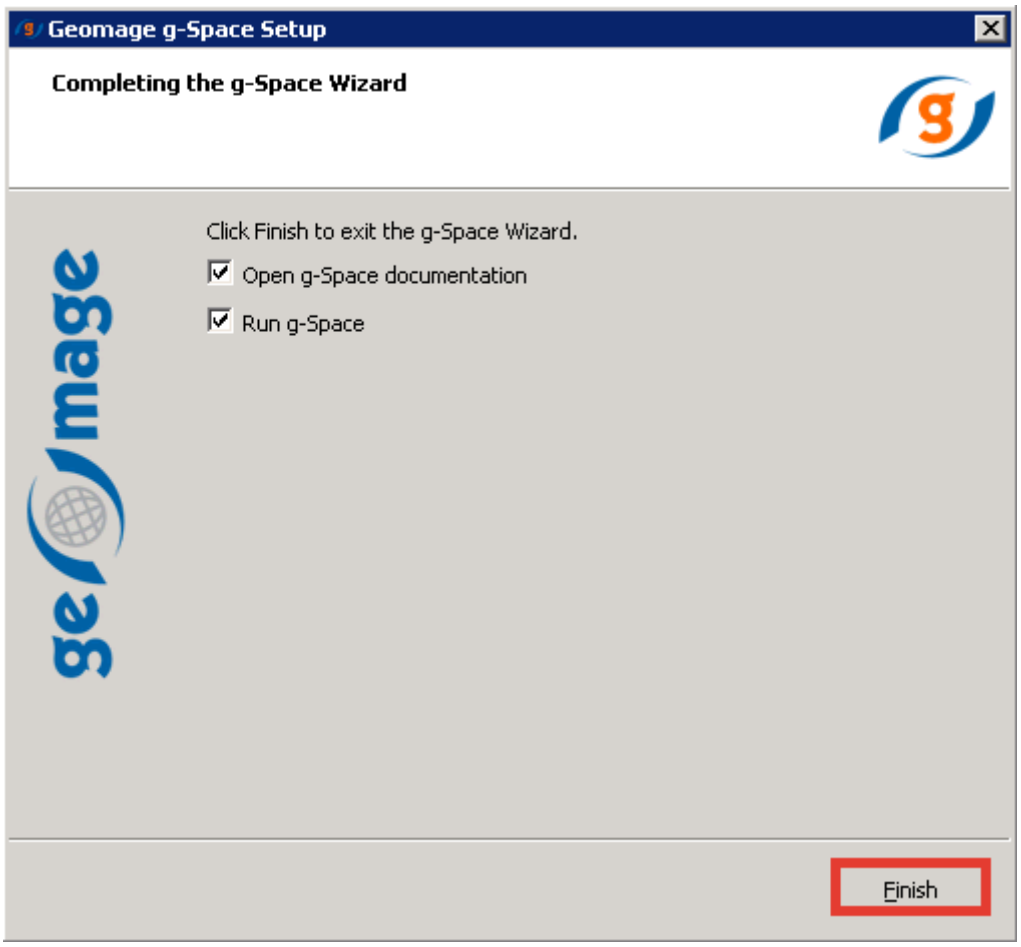
Step 7
It may take few seconds to finish the installation.

© 2026 Geomage 2003 LTD

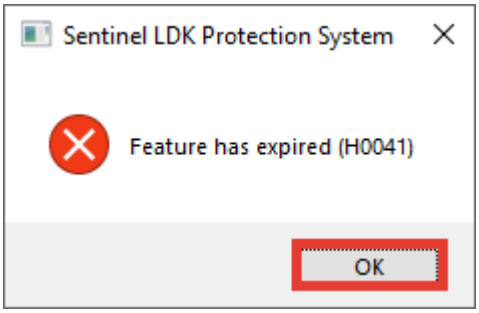


Step 8
Click "Finish" to complete the installation of g-Space. Uncheck **Run g-Space** if you want to run it later.

© 2026 Geomage 2003 LTD



Once you have finished the installation process the message below may appear due to missing license file.



Press OK and follow the instructions in the licensing 23 section

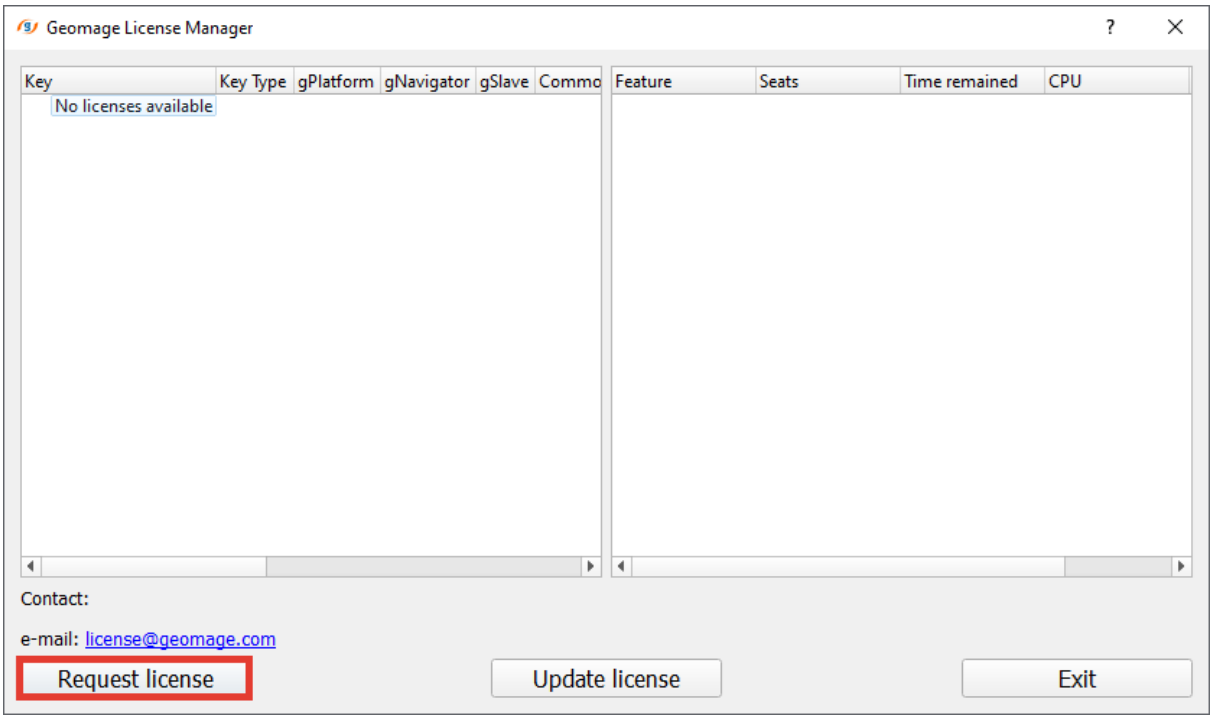
© 2026 Geomage 2003 LTD

2.3 Licensing

To obtain a license for your product, please follow the steps below:

Step 1

- Launch g-Space using the start menu shortcut, or the shortcut installed on your desktop. This will launch Geomage Licence Manager.
- Click the "Request license" button in the Geomage License Manager.

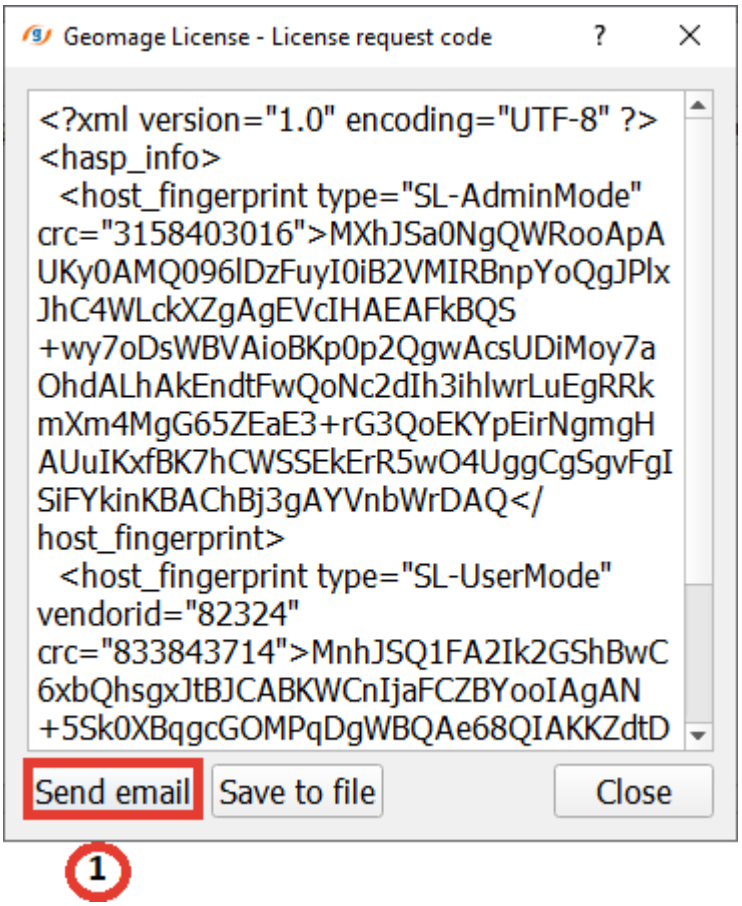


Step 2

Once you have requested a license, the License manager will generate a License request code. You may choose two options:

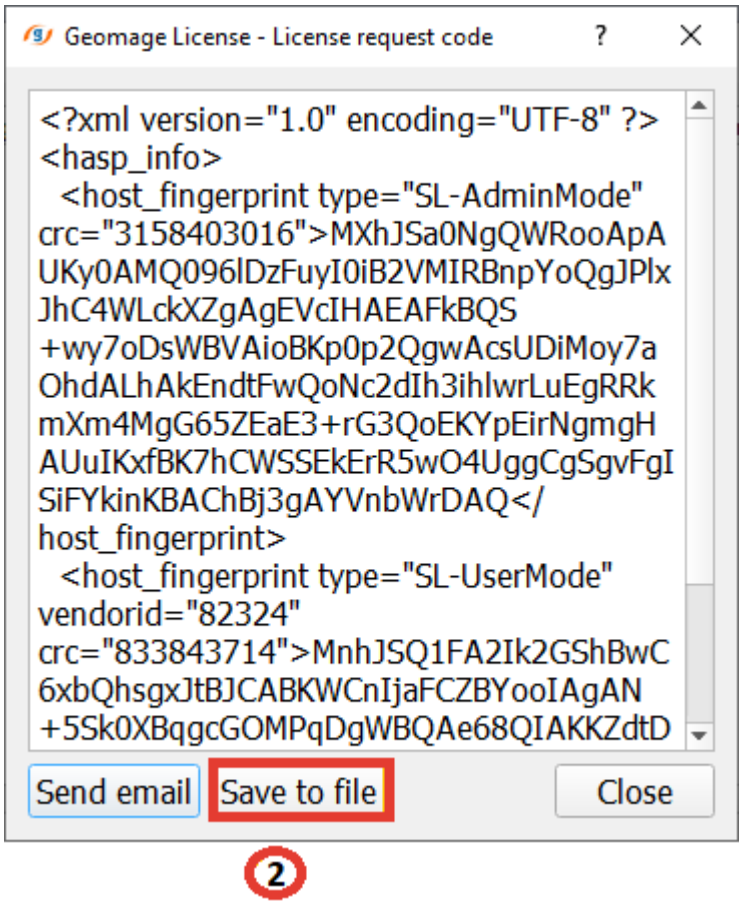
***Option 1*:** Click "Send email" (1) to send the "License request code" to Geomage via your default email client.

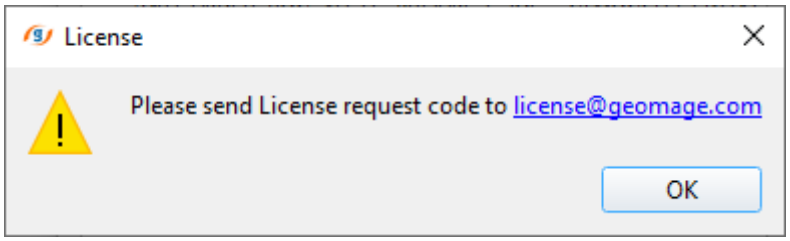
© 2026 Geomage 2003 LTD



***Option 2*:** Copy the "License request code" (2) and send the entire string to **"license@geomage.com"** via a server or other email service.

© 2026 Geomage 2003 LTD





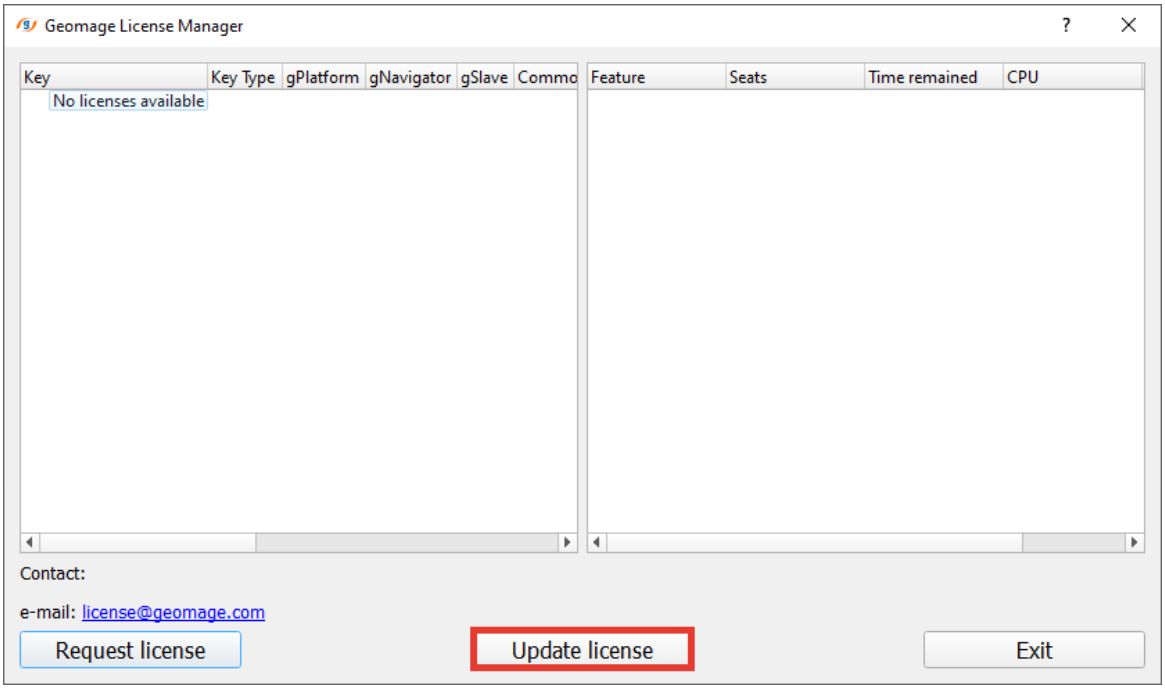
Geomage will then send you a license file via email.

Step 3

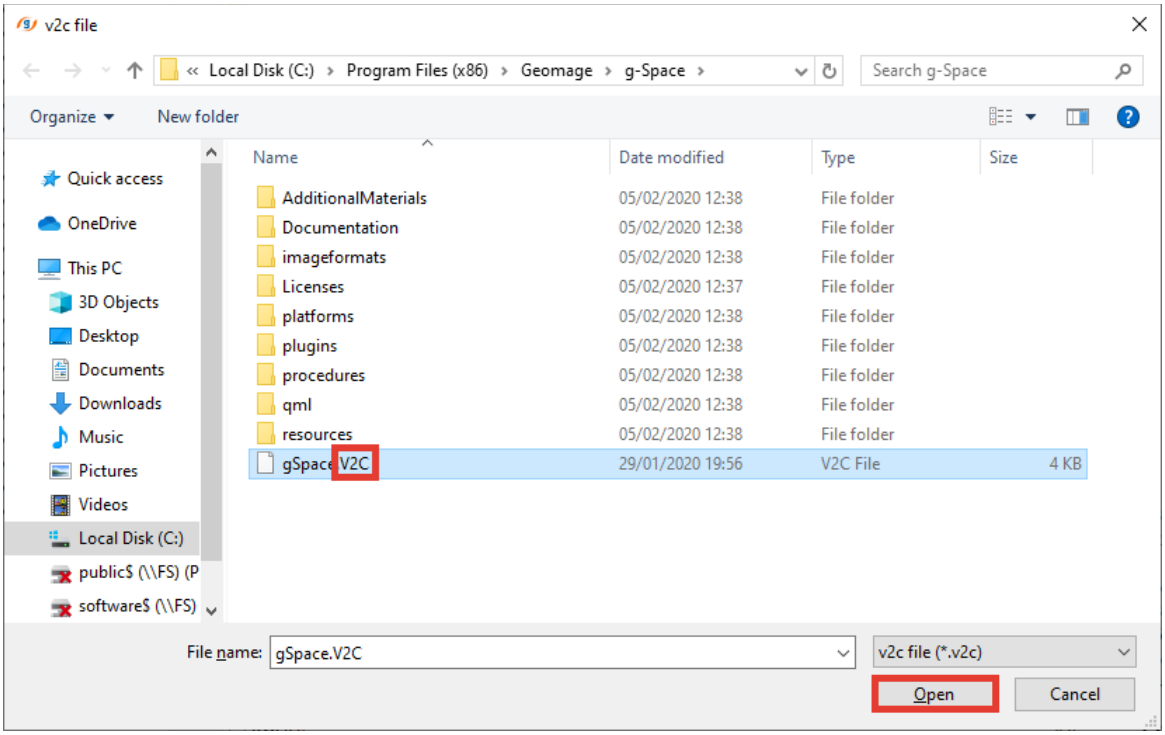
Once you have received your license, you must activate it.

1. Save the received file to a disk on your computer.
2. Go to the Geomage License manager and Click **Update license**.

© 2026 Geomage 2003 LTD



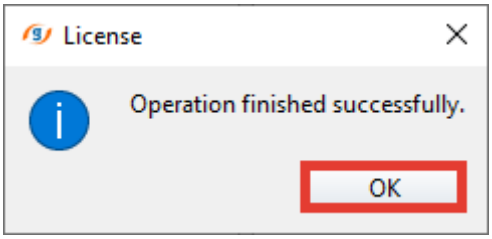
3. Chose the saved file (with suffix .V2C) and execute it by clicking "open".



Step 4

Click **OK** to complete the activation process.

© 2026 Geomage 2003 LTD

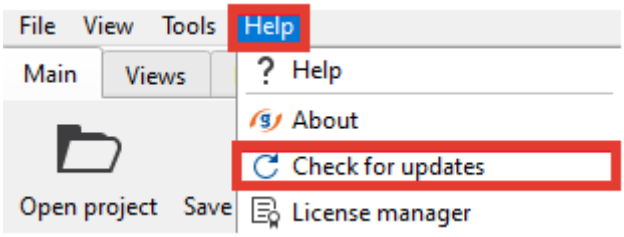


Step 5
Enjoy g-Space

2.4 Software update

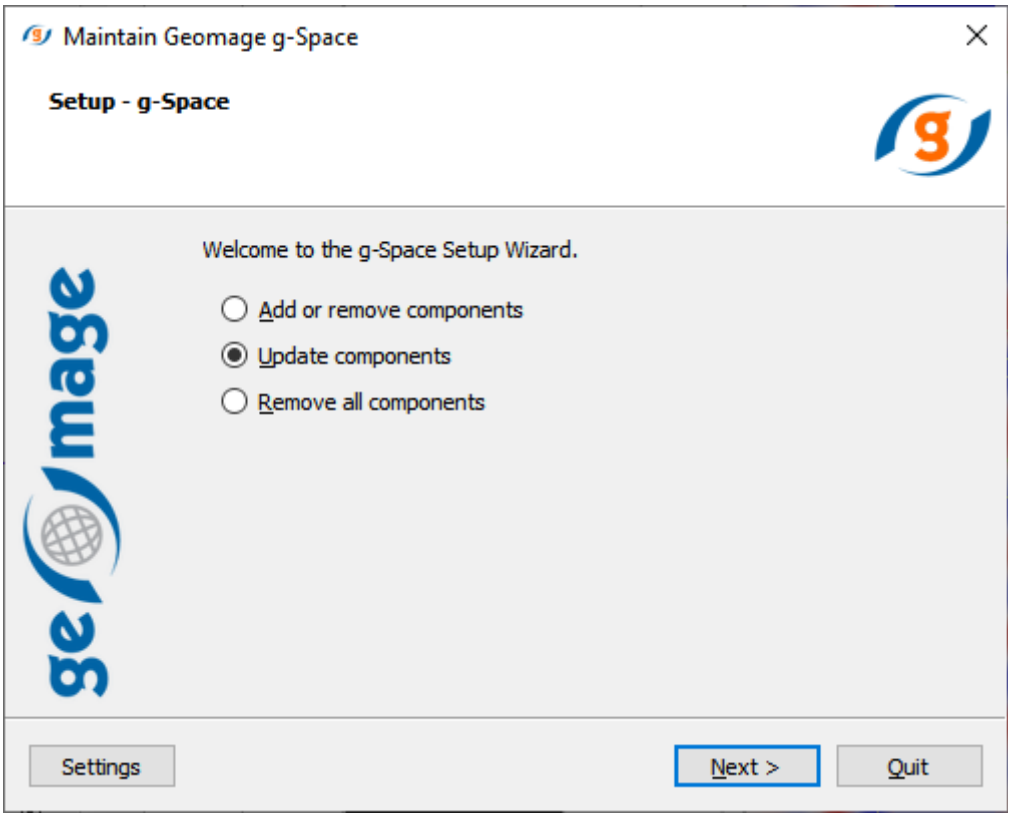
On start of g-Space the application checks (on a daily basis) if a new version was uploaded to repository of Geomage. In case there is a newer version than installed an appropriate message will appear with suggestion to install the update.

In addtion, when needed, User can get the latest version of g-Space by clicking on Help menu and selecting **Check for updates** as shown on the image



That will initiate sequence of installing the product. Please proceed with **Update components** option.

© 2026 Geomage 2003 LTD



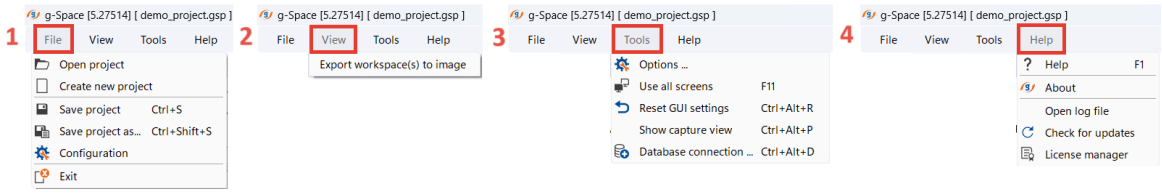
© 2026 Geomage 2003 LTD

User Interface

3 User Interface

Let's start by getting acquainted with the program by exploring the interface.

Subscribe to our channel and watch g-Space interface overview on YouTube



The main menu of the g-Space software includes the following tabs:

1. **File** - This tab provides standard options for project management, such as:
   - **Open project:** Open a previously saved project.
   - **Create new project:** Create a new project.
   - **Save project:** Save the current project.
   - **Save project as...:** Save the current project under a new name or location.
   - **Configuration:** Set configuration settings for the software.
   - **Exit:** Close the application.

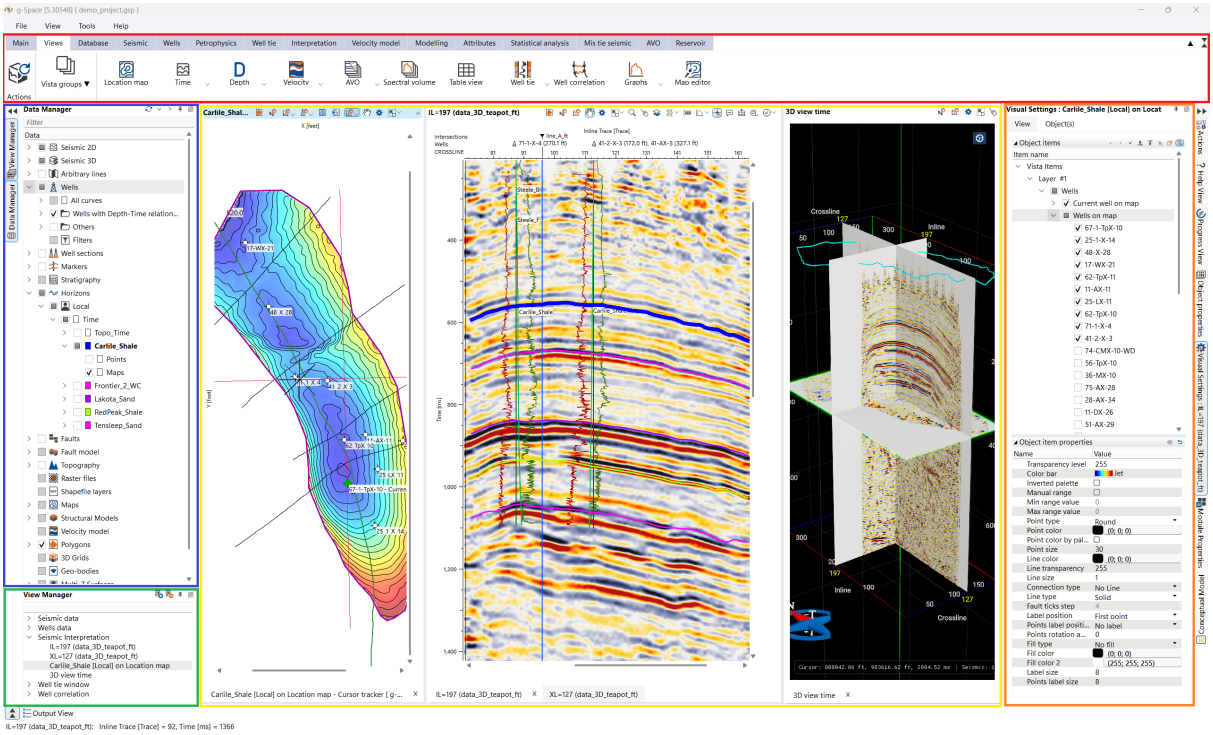
2. **View** - This tab contains options related to the display and export of the workspace:
   - **Export workspace(s) to image:** Export the current workspace(s) as an image file.

3. **Tools** - This tab provides tools for advanced settings and configurations:
   - **Options...:** Open the options menu for further configurations.
   - **Use all screens:** Toggle the use of multiple screens (shortcut: F11).
   - **Reset GUI settings:** Reset the graphical user interface settings to default (shortcut: Ctrl+Alt+R).
   - **Show capture view:** Display the capture view in the workspace (shortcut: Ctrl+Alt+P).
   - **Database connection:** Manage database connections (shortcut: Ctrl+Alt+D).

4. **Help** - This tab offers help and support options:
   - **Help:** Access the help documentation (shortcut: F1).
   - **About:** View information about the software version and credits.
   - **Open log file:** Open the log file to view recent actions and errors.
   - **Check for updates:** Check for available software updates.
   - **License manager:** Manage software licenses.

The g-Space workspace consists of several areas, each of which can be customized individually.

© 2026 Geomage 2003 LTD

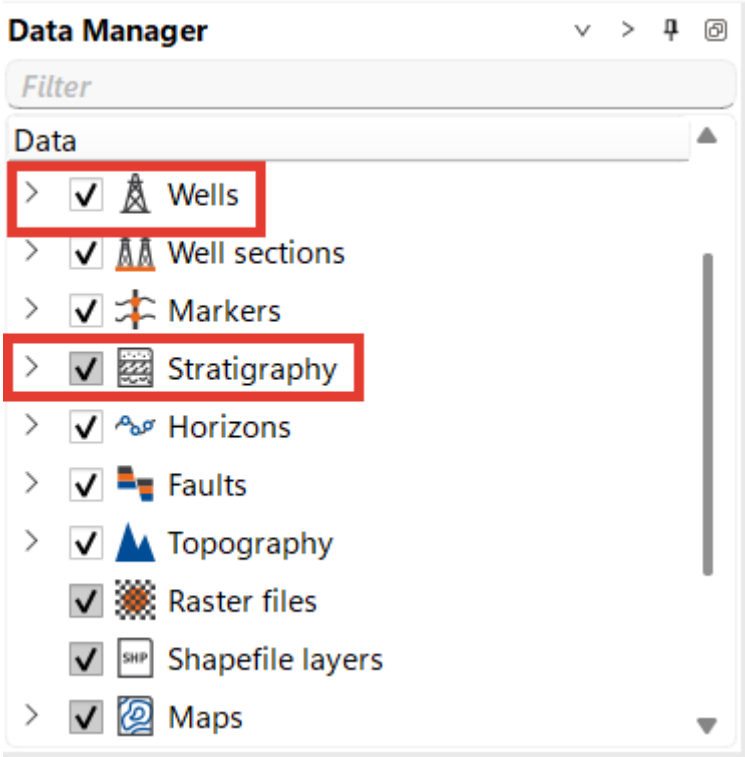


1. Ribbon Bar 33 is the topmost section of the interface, highlighted in the red box. Ribbon Bar is where you can access the main functionalities of the g-Space software. It is organized into several thematic tabs that group related tools and commands.

2. Data Manager 124 Panel is located on the left side of the interface, highlighted in the blue box. Data Manager is a panel used to control and organize the project's database. It displays a hierarchical view of all the data available within the project.

For data that can be visualized in the active window, the display icon will appear transparent. If the data cannot be visualized in the active window, the icon will be highlighted in gray.

© 2026 Geomage 2003 LTD



3. View Manager 189 Panel is located below the Data Manager on the left side, highlighted in the green box. View Manager is responsible for managing the Workspaces 224 and views within the g-Space application. Lists different workspaces and the views within them, such as Seismic data, Wells data, 3D view time, Well tie window, Well correlation, etc. Allows you to control the visibility, arrangement, and management of various views, enabling efficient handling of multiple data displays simultaneously.

Conceptual Model 191 panel is located below the View Manager on the left side. It is used to build a structured geological framework by linking project data to geological objects for use in Structural Modeling.

4. Workspace Area. 224 The central area of the interface, highlighted in the yellow box. Workspace Area is where the main data visualizations and analysis take place. This area is used to display multiple Views 233 that are part of the ongoing workflow or project. Each view within the workspace can represent different types of data, such as seismic sections, well logs, map views, etc.

The cursor position can be synchronized on the map and section.

© 2026 Geomage 2003 LTD

The active window is highlighted in blue as in our Location map example and the Module Properties Panel is displayed for the active window.

5. Module Properties Panel 197 is located on the right side of the interface, highlighted in the orange box. The Module Properties panel allows you to view and modify the properties and settings of the currently selected view or module. Includes adjustable settings such as axes properties, color settings, scaling, grid options, and other parameters specific to the active module. Also on the right side you can switch to other panels, such as

Visual Settings 206 that controls the appearance of the active view and of the objects displayed in it

Progress view 221 that showing progress of the currently running process

Object properties 221 that shows the properties of the object selected in the Data Manager

Actions 214 panel contains processes that can be executed in the software

RAM usage indicator

A memory usage indicator is shown on the right side of the status bar at the bottom of the window. It consists of a small progress bar and a **used / total GB** label that report how much of the computer's physical memory (RAM) is currently in use. Hovering over the bar shows the used and total amounts as a tooltip.

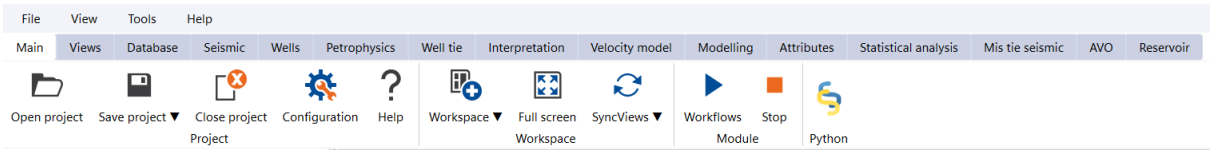
The bar is colour-coded so memory pressure is easy to monitor:

- Green - up to 60% of RAM in use
- Orange - between 60% and 90%
- Red - above 90%

The g-Space interface is designed to be intuitive and flexible, enabling users to easily manage and analyze complex geophysical data. By familiarizing yourself with these components, you can efficiently navigate the software and optimize your workflows.

3.1 Ribbon bar

This chapter will guide through the existing bars available in g-Space and describe the functionality.



© 2026 Geomage 2003 LTD

Every ribbon button shows its name under the icon by default. To change this, open the **Tools** menu, choose **Options ...** and use the **Toolbar buttons** group on the **General** tab. Clear **Show button names** to display the ribbon as icons only, or set **Button width**, **Button height** and **Spacing between buttons** to change the size of the buttons and the gap between them.
The tab names and the section name under each group are always shown, and pointing at a button displays the full command name as a tooltip. Changed ribbon settings are applied the next time g-Space starts.

Main bar 35 - allows to perform the general actions with the project

Views bar 38 - allows to add different functional views 233 in workspace 224

Database bar 41 - manages connections to the g-Space and g-Platform databases for data import, export and synchronization

Seismic bar 44 - allows to add/delete seismic datasets as well as perform functions with loaded seismic data

Wells bar 46 - allows to import/export/delete wells data as well as perform functions with wells

Petrophysics bar 49 - allows to run petrophysical calculations and interpretation tools, providing access to multiple calculators for processing and analysing well log data.

Well tie bar 93 - allows to setup and execute well-tie functionality

Interpretation bar 96 - provides picking controls and horizon and fault interpretation management tools

Velocity model bar 99 - allows to customize your velocity model

Modelling bar 100 - allows to setup the mapping parameters

Attributes bar 105 - allows to calculate/generate attributes (as amplitude, envelope, instantaneous phase etc.) and spectral decomposition

Statistical analysis bar 111 - allows to generate and analyze the cross plots

Mis tie seismic bar 117 - allows to check and fix the seismic intersections misties

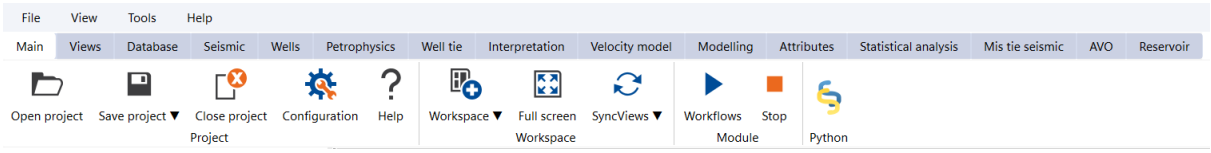
AVO bar 119 - allows to perform AVO analysis

Reservoir bar 122 - allows to make gross rock volume calculation

© 2026 Geomage 2003 LTD

3.1.1 Main bar

Main bar allows to perform the general actions with the project



The Main bar consists of four sections: Project, Workspace, Module and Python. Database operations have their own dedicated Database 41 tab.

Project section:

Open project 361 - opens an existing project. The selected project replaces the one currently open in the same g-Space session; if the current project has unsaved changes, g-Space asks whether to save them before the new project is loaded

Save project 359 / Save project as... On selection g-Space will save all setup and configuration of the project, including all data and views. For seismic data, only paths of seismic data are saved inside the project

Close project 362 - on selection, g-Space will unload open project and create an empty project

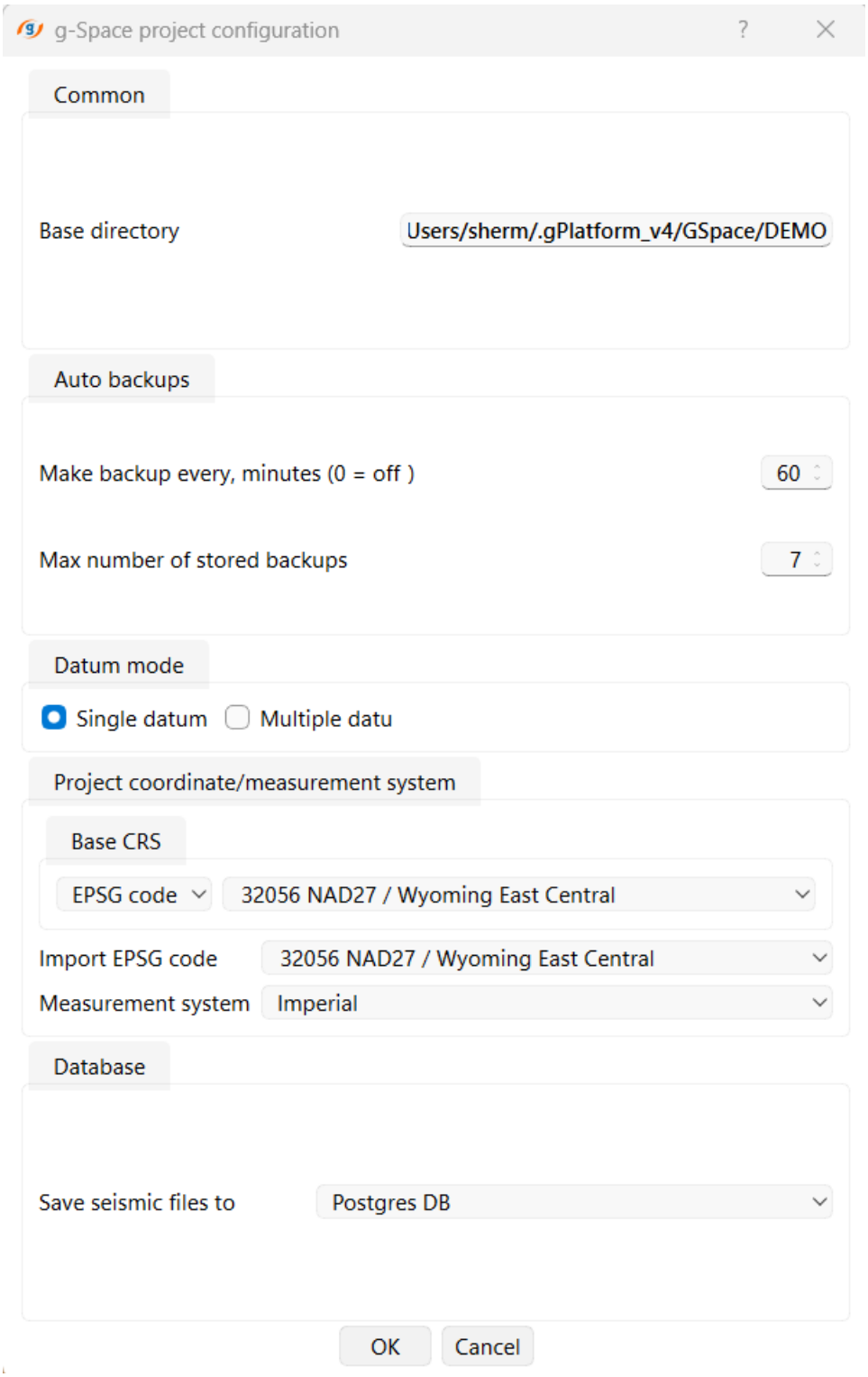
Configuration 363 - opens the project configuration dialog where the global settings of the project are defined

The configuration dialog is organized into the following groups:

- **Base directory** — the root folder of the project. All project files, databases and seismic references are stored relative to this path.
- **Auto backups** — the backup interval in minutes (0 = off) and the maximum number of stored backups. Protects against data loss during long sessions.
- **Datum mode** — choose Single datum or Multiple datum. Single datum applies one reference level to the entire project; Multiple datum allows per-line adjustments.

© 2026 Geomage 2003 LTD

- **Project coordinate / measurement system** — define the Base CRS (EPSG code) and the Import EPSG code for incoming data, and set the measurement system (Metric or Imperial).
- **Database** — select where seismic files are stored: a Postgres database for large-scale projects or file-based storage for simpler setups.



© 2026 Geomage 2003 LTD

Help - opens the g-Space help system

Workspace section:

Workspace 224 - when an arrow button clicked, allows switch between existing workspaces (graphical setups) or in case of clicking on the main button of the combobox allows to create a new empty workspace

Full screen - switch to Full screen mode that allows to put application in full-screen mode along all connected monitors. Shortcut key is F11

Synchronize views 228 - allows to synchronize different views according to their axes. If synchronization selected all corresponding axes from different views will be scrolled and zoomed accordingly. The cursor position will also move synchronically on corresponding views. Views and/or specific axes can be selected/unselected to be synced in the settings dialog that appears when an arrow button of SyncViews clicked

Module section:

Workflows - Opens the Workflows panel to create, edit, and run automated job sequences.

Stop process - interrupts Execution process and might put application state into unstable condition. It's advised to re-run Execution in such case

Python section:

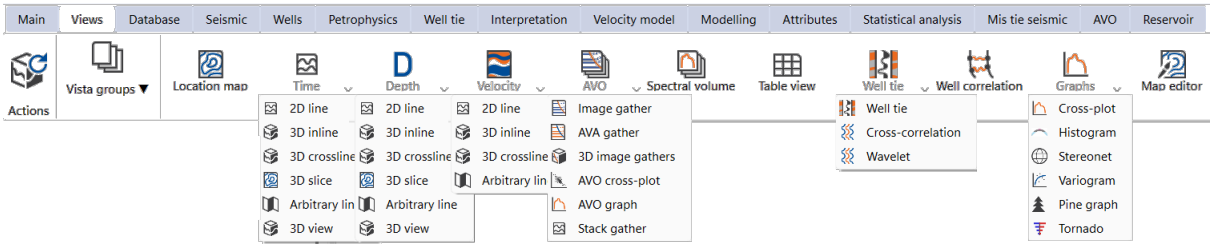
© 2026 Geomage 2003 LTD

The **Python** section provides access to the g-Space Python integration through the **Python calculator** button, which opens the Python calculator for running Python scripts against the project's data. See Python scripting 811 for details.

Database operations: connecting to a g-Space or g-Platform database, importing and exporting data, and synchronizing settings are now performed from the dedicated Database 41 tab of the ribbon bar.

3.1.2 Views bar

Views 233 bar allows to add different functional views 233 into the current workspace 224.



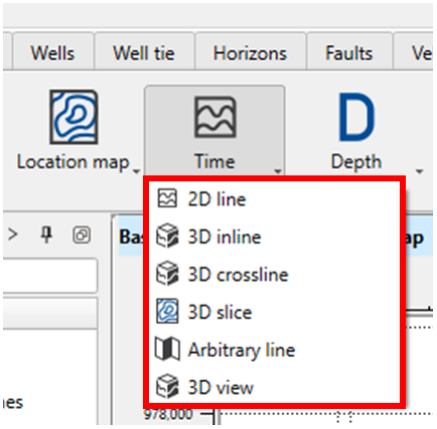
Actions - Update 3D view

Vista groups - specific views which are not presented as buttons on the right

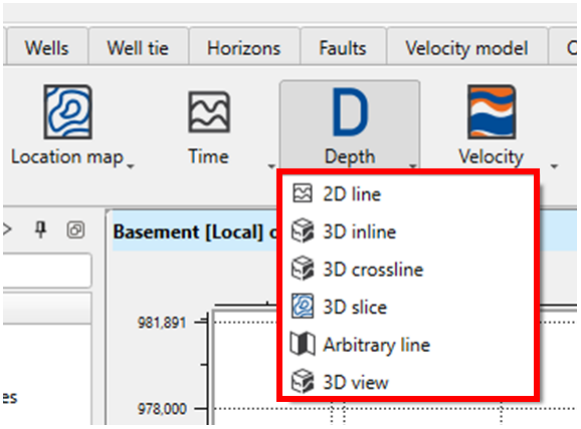
Location map 235 - creates map view with ability to show and control loaded GIS-type data, such as seismic, well data geometry, polygons, arbitrary lines, maps, horizons, etc. Clicking the button opens a new, empty Location map view.

Time 240 - allows to add different seismic views in time domain (2D line, 3D inline, 3D crossline, 3D slice, Arbitary line, 3D view):

© 2026 Geomage 2003 LTD

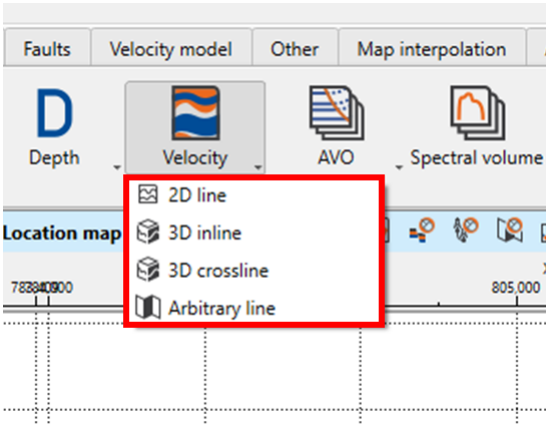


Depth - allows to add different seismic views in depth domain (2D line, 3D inline, 3D crossline, 3D slice, Arbitary line, 3D view):

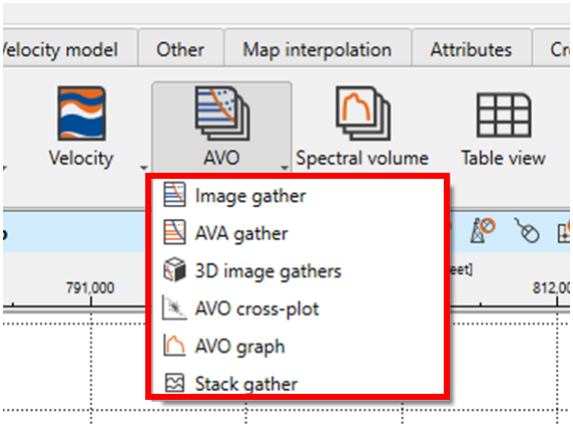


Velocity - allows to add different velocity views in depth domain that were created by Velocity model building flows or loaded from external sources (2D line, 3D inline, 3D crossline, Arbitary line):

© 2026 Geomage 2003 LTD



AVO 272 - allows to add different views for AVO analysis (Image gather, AVA gather, 3D image gather, AVO cross-plot, AVO graph, Stack gather):



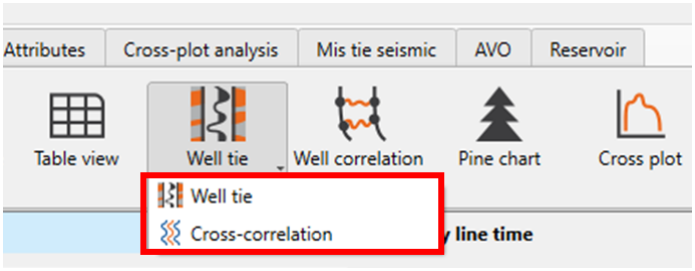
Spectral volume - creates view for spectral analysis

Table view 285 - allows to create table view for review and edit different types of tabular format data

Well tie 287 - contains two views. Well tie view allows to visualize, analyze and edit loaded checkshot, generate synthetic and perform seismic to well tie flow.

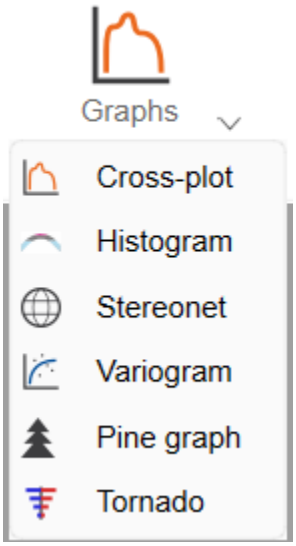
Cross-correlation view visualizes cross-correlation function between seismic and synthetic traces:

© 2026 Geomage 2003 LTD



Well correlation 296 - creates a view for correlation and visualization of different types of well data such as continuous and/or discrete log data, point data, comment logs, etc.

Graphs 326 - creates views for statistical analysis data, consisting of Cross-plot 745, Histogram 748, Variogram 335, Stereonet 756, Pine graph 332, and Tornado 338.



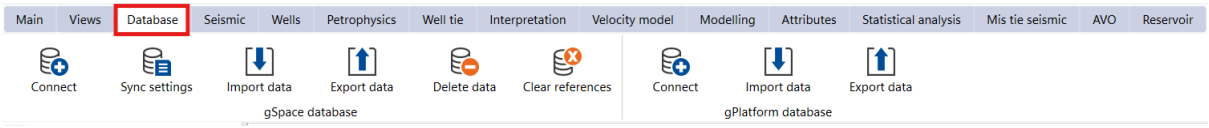
Map editor 341 - creates a view for map editing

3.1.3 Database

The **Database** ribbon bar exposes the project's connections to remote object stores. It is split into two sections: **gSpace database** for the native g-Space multi-user server, and **gPlatform**

© 2026 Geomage 2003 LTD

database for connections to Geomage's gPlatform data service. Each section offers its own connect, import and export buttons.



gSpace database

Operations that act on the native g-Space project database (used for multi-user work).

- **Connect** — opens the **Database connection** wizard where the user supplies the server address and credentials needed to attach the current project to a remote gSpace repository. See Multi-user mode 795 for the full workflow.
- **Sync settings** — opens the **Sync settings wizard** described below; selects which project objects participate in database synchronisation.
- **Import data** — fetches objects (wells, horizons, seismic, polygons, ...) from the connected gSpace database into the current project.
- **Export data** — uploads local objects back to the gSpace database; the same dialog is used by the per-object **Send to DB** context action.
- **Delete data** — removes selected objects from the remote database.
- **Clear references** — breaks the link between local objects and their remote counterparts without touching the remote data. Useful when a project is forked into a standalone copy.

gPlatform database

Operations that act on Geomage's hosted gPlatform data service.

- **Connect** — opens the **Connect to gPlatform database** wizard described below.
- **Import data** — fetches datasets from gPlatform into the current project.
- **Export data** — uploads selected project objects to the configured gPlatform account.

Connect to gPlatform database

This dialog captures the connection parameters used for every subsequent gPlatform import/export. The wizard validates the input before allowing the project to store credentials.

Input fields:

- **Hostname** — DNS name or IP address of the gPlatform server.
- **Port** — TCP port used by the gPlatform endpoint.
- **Database** — name of the target database on the server.
- **Username** / **Password** — account credentials.

© 2026 Geomage 2003 LTD

Actions:

- **Test Connection** — tries to reach the server with the supplied parameters and reports success or failure without saving them.
- **Connect** — stores the credentials in the project and opens the connection for the current session. Invalid input fields blink red until corrected.

Sync settings wizard

The Sync settings wizard controls the scope of database synchronisation. It lists every syncable object in the project (horizons, faults, wells, polygons, maps, ...) and lets the user pick which ones to push or pull.

Main controls:

- **Search field** — filters the object tree by name.
- **Object tree** — hierarchical list grouped by data type (Horizons, Faults, Wells, Polygons, Maps, ...), each entry with a checkbox.
- **State filters** — show only objects in a given sync state: **Created**, **Modified** or **Deleted**.
- **Total size indicator** — estimates how much data the current selection represents before uploading.
- **Context menu** — batch check / uncheck all objects of a given type.

Object revision list

Most database-linked objects expose an **Object revision list** command in their context menu. It opens a table summarising every revision of the selected object stored on the server, with six columns: timestamp, user, revision ID, object state, comment (editable) and action.

Common operations:

- Edit a comment in-place to document why a revision was produced.
- **Rollback** — revert the local object to a previous revision.
- Export the visible history through the window's toolbar to share an audit trail with other users.

See Also

- Multi-user mode 795
- Import data 373
- Export data 464
- Data manager 124

© 2026 Geomage 2003 LTD

3.1.4 Seismic bar

Seismic bar allows you to add/delete seismic data sets in the project

For information about the process of loading seismic data into the project, refer to chapter Import seismic 373

The Seismic bar consists of the following sections: Views, Add, Clear, Management, Calculator and Trace header dump.

Views section:

Open predefined seismic data 228 workspace

Workspace 224: arrow button allows to switch between existing workspaces. Main button of the combobox allows to create a new empty workspace

Add section:

Add dataset - adding single dataset to the project. After adding data it still have to be reviewed and loaded into the project

Add and load dataset - adding and loading a single seismic dataset to the project. After adding data by that operation, it's ready to be used within the project. It assumes that all data is correct as loaded

Load dataset - loading a single seismic dataset that was previously added to the list of project's seismic. It assumes that all needed QC was already done

© 2026 Geomage 2003 LTD

Load all datasets - loading all seismic datasets that was previously added to the list of project's seismic or were changed during QC process. It assumes that all needed QC was already done on all seismic datasets

Clear section:

Clear dataset - remove a single selected seismic dataset from the project. The application does not deletes data from the disk

Clear all datasets - remove all seismic datasets from the project. The application does not deletes data from the disk

Management section:

Create seismic group

Propagate current shift on whole line

Calculate datum level

Import seismic data from ASCII. See more information in Convert ASCII to seismic 384

Calculator section:

Opens the seismic calculator, used to perform arithmetic operations between seismic datasets and create new calculated volumes.

© 2026 Geomage 2003 LTD

Trace header dump section:

Reads and exports trace header values from the selected seismic dataset. Set the header **Format** (for example Int4) and the **1st byte** position, then run the dump to inspect the header content. See also Trace headers math 521.

3.1.5 Wells bar

Wells bar allows to import/export/delete well data of the project

For more information about the formats supporting by g-Space please refer to chapter Well Data Formats 507

Consists of the following sections: Views, Log mode, Headers, Markers, Checkshots, Well inclinometry, LOG, Comments, Calculator, Wells visualisation and Templates

Views section:

Open predefined wells data 232 workspace

Workspace 224 - arrow button allows to switch between existing workspaces. Main button of the combobox allows to create a new empty workspace

Log mode section:

Log Mode - a drop-down that controls how logs are displayed in the Data Manager, switching between two modes. In **Curves** mode (the default) logs are shown as individual curves; in **LAS files** mode logs are grouped by the original imported LAS file.

© 2026 Geomage 2003 LTD

Headers section:

Import 393 / **Export** 464 / Clear Headers - operations on well headers information

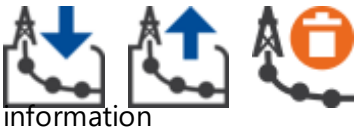
Markers section:

Import 403 / **Export** 468 - operations on well markers data

Checkshots section:

Import 414 / **Export** 471 - operations on time-depth relationship data within each well

Well inclinometry section:



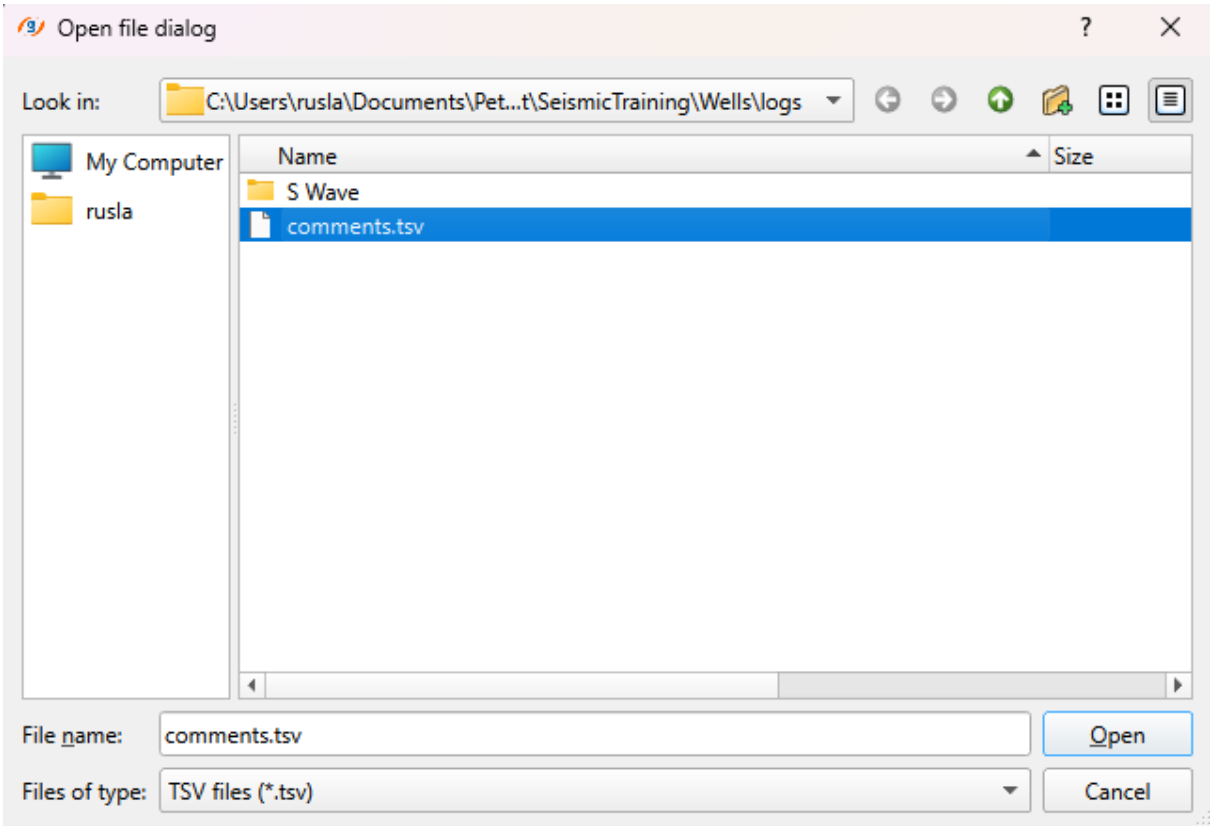
Import 397 / **Export** 466 - operations on well deviation information

LOG section:

Load LAS files 407 / **Log Loading Wizard** / **Export LAS files** - operations on well log data

Comments section:

© 2026 Geomage 2003 LTD



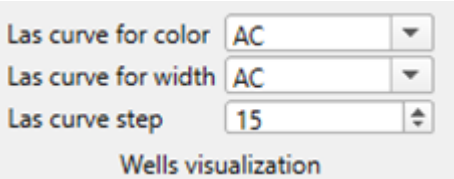
Import 421 / **Export** 477 - operations on well log data

Calculator section:

Markers attributes calculator - opens the calculator used to derive attributes from well markers (for example thickness or depth differences between markers). See also Working with Markers 551.

Wells visualisation section:

© 2026 Geomage 2003 LTD



Settings for 3D visualization

It contains three options:

1. **Las curve for color**: A drop-down menu where the user can select a LAS curve color.
2. **Las curve for width**: A drop-down menu where the user can select a LAS to define the width of the Log curve in the 3D view.
3. **Las curve step**: A numeric input field set to 15, which specifies the step value for the log data used for visualization.

Templates section:

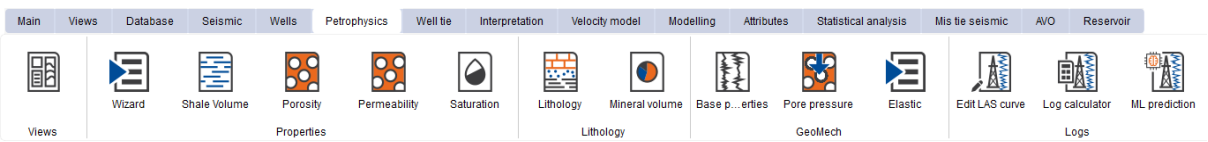
Select template - a drop-down of saved templates that applies the chosen template to the active well view: to the **Well log view** when one is active, or to the Well correlation view 296 (well sections) when that view is active. If no compatible view is open, g-Space shows the message *Open a Well log view or Well correlation view to apply a template*. The template actions in the Well sections 150 context menu remain available.

Manage well templates - opens the well-template manager to rename or delete saved well view templates. To create a template, use **Save Template** in the applicable well view.

Note: Clear actions (Clear LAS files, Clear checkshot, Clear wells, Clear markers) are available on the Data Manager right-click (context) menu, not on the ribbon.

3.1.6 Petrophysics

The **Petrophysics bar** allows you to execute petrophysical calculations such as shale volume, porosity, saturation, pore pressure, and more using well log data available in the project.



© 2026 Geomage 2003 LTD

For more information about input formats and methods used in each calculator, refer to the dedicated chapters for each calculation module.

The Bar consists of 5 sections: **Views, Properties, Lithology, GeoMech, Logs.**

Views section:

Open predefined petrophysics calculation workspace.
The workspace includes the following views by default:

- Well correlation view 296
- Cross-plot view 330
- Well log histogram 327

Properties, Lithology, GeoMech and Logs sections:

These sections include the calculators that operate on well log data. Each button opens a wizard or dialog for the selected operation.

Properties: **Petrophysics wizard**, **Shale Volume**, **Porosity**, **Permeability**, **Saturation**. **Lithology**: **Lithology interpretation**, **Mineral Volume Estimation**. **GeoMech**: **Base properties**, **Pore pressure**, **Elastic properties**. **Logs**: **Edit LAS curve**, **Log calculator**, **ML Log prediction**. On the buttons the longer names are shortened to **Wizard**, **Lithology**, **Mineral volume**, **Elastic** and **ML prediction**, and a caption that is too wide for its button is cut in the middle, for example **Base p...erties**.

These wizards are modeless: the main window stays available while a wizard is open, so you can keep using the ribbon, the views and the Data manager 124, and review the results in the views between **Run** cycles without closing the wizard.

Petrophysics wizard – Opens the **Unified Petrophysics** wizard, which carries the whole interpretation in one dialog instead of one calculator at a time. Its tabs follow the calculation order: **1. Environment**, **2. Vsh**, **3. PHI**, **4. Saturation**, **5. Permeability**, **6. Matrix** and **7. Results**.

Base properties – Opens the base property calculation wizard. This module allows estimation of hydrostatic pressure, temperature gradient, overburden stress, and effective stress using well depth and fluid parameters. Input parameters include fluid

© 2026 Geomage 2003 LTD

density, gravity constant, and depth range. The resulting curves can be saved as new well logs for further use in pore pressure or stress analysis.

Shale Volume calculation

Shale Volume – Opens the shale volume calculation module. Select the method (e.g., linear, Larionov) and input the gamma ray or other relevant logs.

Porosity calculation

Porosity – Opens the porosity calculation wizard. Supports various methods (density, sonic, neutron, or combined) and requires input of matrix and fluid properties.

Permeability – Opens the permeability calculation wizard. Its methods are grouped as **Porosity - Swirr (empirical)** (Timur, Tixier, Morris-Biggs, Wyllie-Rose), **NMR** (Coates, SDR) and **Core-calibrated** (k-φ transform, Kozeny-Carman, Winland R35), and the output curve is named after the selected method, for example *Permeability - Coates*.

Saturation calculation

Saturation – Opens the saturation wizard. Allows configuration of water saturation methods such as Archie or Simandoux, and includes parameter input for resistivity logs, cementation exponent, and formation water resistivity.

Mineral Volume Estimation – Opens the mineral volume estimation wizard, which estimates the volume fractions of the rock-forming minerals from the well-log responses.

© 2026 Geomage 2003 LTD

Pore pressure calculation **Pore pressure** – Opens the pore pressure estimation module. Includes methods like Eaton's and Bowers', allowing you to define input logs (e.g., velocity, resistivity) and calibration parameters.

Elastic properties – Opens the elastic properties calculation wizard. It calculates **Velocities** (Vp, Vs, Vp/Vs and predicted shear slowness), **Impedances** (acoustic and shear), **Elastic moduli (dynamic)** (Poisson's ratio, Young, shear, bulk and P-wave moduli, Lamé's first parameter and bulk compressibility) and **AVO attributes** (Lambda-Rho and Mu-Rho) from the sonic and density curves of the well. The shear source is selected as **DTS curve**, **Predicted from Vp** or **Vp/Vs constant**.

Lithology interpretation – Opens the lithology interpretation tool. This module enables defining lithotypes (e.g., sand, shale, limestone) based on log value ranges and logical rules. Users can set multiple lithotypes, assign weights to log parameters, define matching thresholds, and manage conflicts between overlapping definitions. The result is a lithology log representing the dominant lithotype per depth interval.

Edit LAS curve – Opens the LAS curve editing dialog. Provides options to modify existing well log curves using filters (moving average, median, bandpass), apply depth shifts, or perform mathematical operations with expressions. Includes a history panel for tracking applied modifications and undoing changes.
For more information refer to this topic: LAS editing 320

LOG calculator Opens well log calculator. This tool is used to generate new well logs by applying mathematical expressions to existing logs. It enables the creation of customized well log data for further analysis.

For more information about the well log operations please refer to Well log calculator 558

ML Log prediction **ML Log prediction** Opens Machine Learning based wizard that enable Log data prediction calculation

© 2026 Geomage 2003 LTD

Common calculator settings

The calculators in these sections share the same input layout, unit handling and run reporting:

- **Method** – when a method is selected, the wizard keeps only the input curves and parameters that this method uses and hides the rest.
- **Input curves** – each curve list contains all curves of the selected well. A curve field is pre-filled with a curve whose log type matches the required log, and is left empty when the well has no curve of that type.
- **Units** – every curve and every numeric parameter has its own unit list or unit label, and the default follows the project measurement system: density in g/cc (Metric) or lb/ft³ (Imperial), with kg/m³ also selectable; travel time in µs/m or µs/ft; and proportional values as fraction or %.
- **Batch calculation** – the adjacent list offers **All wells** and, in addition, the saved well filters, well folders and well sections of the project, so a single run can be limited to one of these groups.
- **Depth range** – **Base properties**, **Shale Volume**, **Porosity**, **Saturation**, **Mineral Volume Estimation**, **Pore pressure**, **Lithology interpretation** and **Edit LAS curve** limit the calculated interval on the same tabs: **MD range**, **TVD range**, **TVDSS range** and **Markers range**. **Start** and **End** are one pair of boundaries shared by these tabs, so selecting **Start** or **End** in one tab selects it in all of them, and the same boundary is only expressed as measured depth, true vertical depth, depth below sea level or a marker. The check boxes carry the current depth unit, for example **Start (m)** and **End (m)** in a Metric project and **Start (ft)** and **End (ft)** in an Imperial one.
- **Full depth by default** – a boundary whose check box is cleared keeps its field disabled and shows the full depth of the selected well: **Start** the shallowest and **End** the deepest point of the well trajectory, with the matching values on the **TVD range** and **TVDSS range** tabs, while the **Markers range** lists stay **Undefined**. Selecting another well refreshes these values. Leaving **Start**, **End** or both cleared means that the calculation covers the full available depth of the well, and with **Batch calculation** every well is calculated over its own full interval, so a run no longer has to be prepared by typing depth values by hand (a well that has no trajectory data is skipped). The same defaults apply when a **Base properties** task is re-run from the **Workflow** panel. Only the **Base properties** wizard checks the entered values: if both **Start** and **End** are selected and **Start** is not shallower than **End**, **Run** reports *Start depth must be shallower than End depth.* and the calculation does not start.
- **Zones range** – **Shale Volume**, **Porosity**, **Saturation**, **Permeability**, **Mineral Volume Estimation**, **Pore pressure**, **Elastic properties** and **Lithology interpretation** carry one more tab, which limits the calculation to the selected zones of a zone set. It lists only the zones of the default marker dataset, so zones built on another marker dataset, and

© 2026 Geomage 2003 LTD

zones whose marker dataset has been removed from the project, are not offered; to calculate against them, make their dataset the default one with **Set default** in the Data manager. When the project holds zones but none of them belong to the default dataset, the tab reports *No zones are available for the default marker dataset "<marker dataset>"*., and when the project holds no zones at all it reports *No marker zones are available in the project*.

- **Input/Output** – the preview panel carries this title followed by the name of the selected well, and plots the input curves of that well together with the calculated **Output** curve.
- After a run the wizard reports for how many wells the calculation completed and lists them. A well that is missing a required input curve is reported separately together with the missing input and is skipped, so no empty output curve is created for it.

Workflow integration

The petrophysical wizards on this bar can be added to the project Workflow 535 and re-run as tasks. This applies to **Base properties**, **Shale Volume**, **Porosity**, **Saturation**, **Mineral Volume Estimation**, **Pore pressure**, **Lithology interpretation**, **Permeability**, **Elastic properties**, **Edit LAS curve** and **Log calculator**. Each of these wizards has the same row of bottom buttons:

- **Save to workflow** – stores the wizard and all of its settings (selected wells, input and output curves, method, method parameters, user-defined constants and output names) as a Workflow task.
- **Run** – executes the calculation with the current settings (this button replaces the former *Calculate* button).
- **Save** – stores the result in the well. An existing curve with the same output name is replaced, after the confirmation described below.
- **Close** – closes the wizard.

In the **Base properties**, **Shale Volume**, **Porosity**, **Saturation**, **Pore pressure** and **Lithology interpretation** wizards, **Save** replaces a curve that already carries the output name instead of adding a second curve. Whenever at least one of the target wells already contains such a curve, a **Confirm** message appears before anything is written: *The well "<well name>" already contains a curve with that name. Do you want to overwrite it?* for one well, or *Some wells already contain a curve with that name. Do you want to overwrite it?* when several wells are affected.

Yes, which is preselected, replaces the existing curve in every affected well. **No** cancels the save, so nothing is written and the wizard stays open with the result unsaved. If the existing curve belongs to a LAS file that cannot be edited, the save is not completed and reports that the curves cannot be removed from uneditable LAS files.

© 2026 Geomage 2003 LTD

Mineral Volume Estimation does not show this message: it handles existing curves with its own **Curve overwrite** setting (see Mineral Volume Estimation 88).

Log calculator writes the calculated curve to the wells with either **Run** or **Save**, and it replaces a curve that already carries the output name after its own confirmation, which names the curve and the well and can be answered for all remaining wells at once. When a saved **Log calculator** task is executed by a Workflow 535 run, the existing curve is overwritten without a confirmation (see Well log calculator 558).

Reopening a task from the Workflow 535 panel restores the wizard with its saved settings, so they can be reviewed, edited, run, and saved back to the task.

A saved zone selection is checked against the project first: zones whose marker dataset has been removed, and zones that belong to a marker dataset that is not the default one, are dropped from the restored selection. **Shale Volume**, **Porosity**, **Saturation**, **Mineral Volume Estimation**, **Pore pressure** and **Lithology interpretation** report this as *Some zones saved in this workflow are no longer available in the project and were removed from the calculation setup.* and *Some zones saved in this workflow belong to a marker dataset that is not the default one and were removed from the calculation setup. Set that dataset as default to use them.* (in **Lithology interpretation** the wording is *interpretation setup*); **Permeability** and **Elastic properties** drop such zones without a message.

3.1.6.1 Base properties

The **Base properties** module provides a set of fundamental calculations commonly used in petrophysical and geomechanical analysis.
Each tab represents an individual calculator that generates a new log curve for the selected well or a batch of wells.
The module allows users to specify input parameters, define a calculation interval, and visualize the resulting curve before saving it to the project.
Use **Run** to calculate the current tab, **Save** to store its result, or **Save to workflow** to add the configured calculation to the project Workflow 535.

Hydrostatic Pressure

Calculates the hydrostatic pressure as a function of depth using the selected fluid density.

© 2026 Geomage 2003 LTD

The relationship is defined by:

$$Phyd(z) = \rho w \cdot g \cdot z$$

where

- ρw – fluid density,
- g – gravitational acceleration,
- z – depth.

Input parameters:

- Fluid density (g/cc or lb/ft^3)
- Gravity constant (m/s^2 or ft/s^2)
- Measured depth range and step

Output: Hydrostatic pressure curve (MPa or psi)

Temperature Gradient

Computes the temperature distribution with depth using a constant surface temperature and geothermal gradient.
The formula applied:

$$T(z) = Tsurface + G \cdot z$$

where

- $Tsurface$ – surface temperature,
- G – geothermal gradient,
- z – depth.

Input parameters:

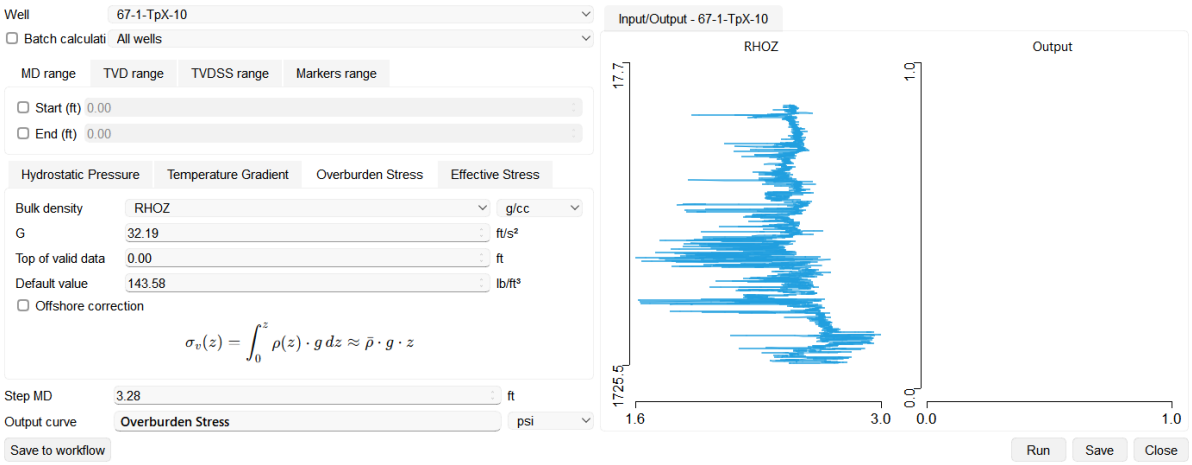
- Surface temperature (°C or °F)
- Temperature gradient (°C/m or °F/100ft)
- Depth step

Output: Temperature Gradient curve

Overburden Stress

© 2026 Geomage 2003 LTD

Estimates the vertical stress resulting from the weight of the overlying sediments.
The calculation integrates the rock bulk density curve over depth:



With **Offshore correction** enabled, the wizard uses *σv(z) = ρw · g · zw + ∫zw^z ρ(z) · g dz*, where ρw is **Water density** and zw is **Water depth**.

where

- *ρ* – bulk density,
- *g* – gravitational acceleration.

At each output depth the result uses the average bulk density from the top of the well down to that depth. Above **Top of valid data** the **Default value** density is used in place of the log. When you press **Run**, the **Default value** density also replaces any depth at which the bulk density log has no valid value, so intervals without a valid reading are still included in that average and the result curve stays continuous.

Input parameters:

- Bulk density log and unit (kg/m³, g/cc, or lb/ft³); default and water-density values use g/cc in Metric projects or lb/ft³ in Imperial projects
- Gravity constant
- Depth interval and step
- **Top of valid data** and **Default value** (the bulk density used above that depth)

Output: Overburden stress curve (MPa or psi)

Effective Stress

© 2026 Geomage 2003 LTD

Determines the effective stress by subtracting pore pressure from total overburden stress:

$\sigma eff = \sigma v - Pp$

where

- σv – overburden stress,
- Pp – pore pressure.

Input parameters:

- Overburden stress curve
- Pore pressure curve

Output: Effective stress curve (MPa or psi)

3.1.6.2 Shale Volume

The **Shale Volume** wizard estimates the fraction of shale within a formation based on well log data. It supports several industry-standard methods using **Gamma Ray (GR)** or **Spontaneous Potential (SP)** logs, as well as a **Neutron–Density** cross-plot method.

The output is a **Vshale (Vsh)** curve that represents the volumetric proportion of shale along the wellbore.

Input Parameters

- **Well selection:** Choose one well or activate **Batch calculation** to process all wells or a selected group from the Data Manager.
- **Depth filter:** Define the interval for calculation by using **MD range**, **TVD/TVDSS range**, or by selecting the depth between **markers**.
- **Curve selection:** Select the log(s) to be used — Gamma Ray or SP for the single-log methods, or Neutron Porosity and Density for the Neutron–Density method.
- **Parameter limits:** Set clean and shale reference values for the chosen curve (manual or automatic).
- **Output curve:** Define the name for the resulting shale volume log.

© 2026 Geomage 2003 LTD

Each method generates a **Vshale (Vsh)** log, displayed alongside the input curve for visual comparison.
Users can review results on the **Input/Output** panel, adjust parameters, **rename the output curve directly in the wizard**, and click **Save** to store it in the project.

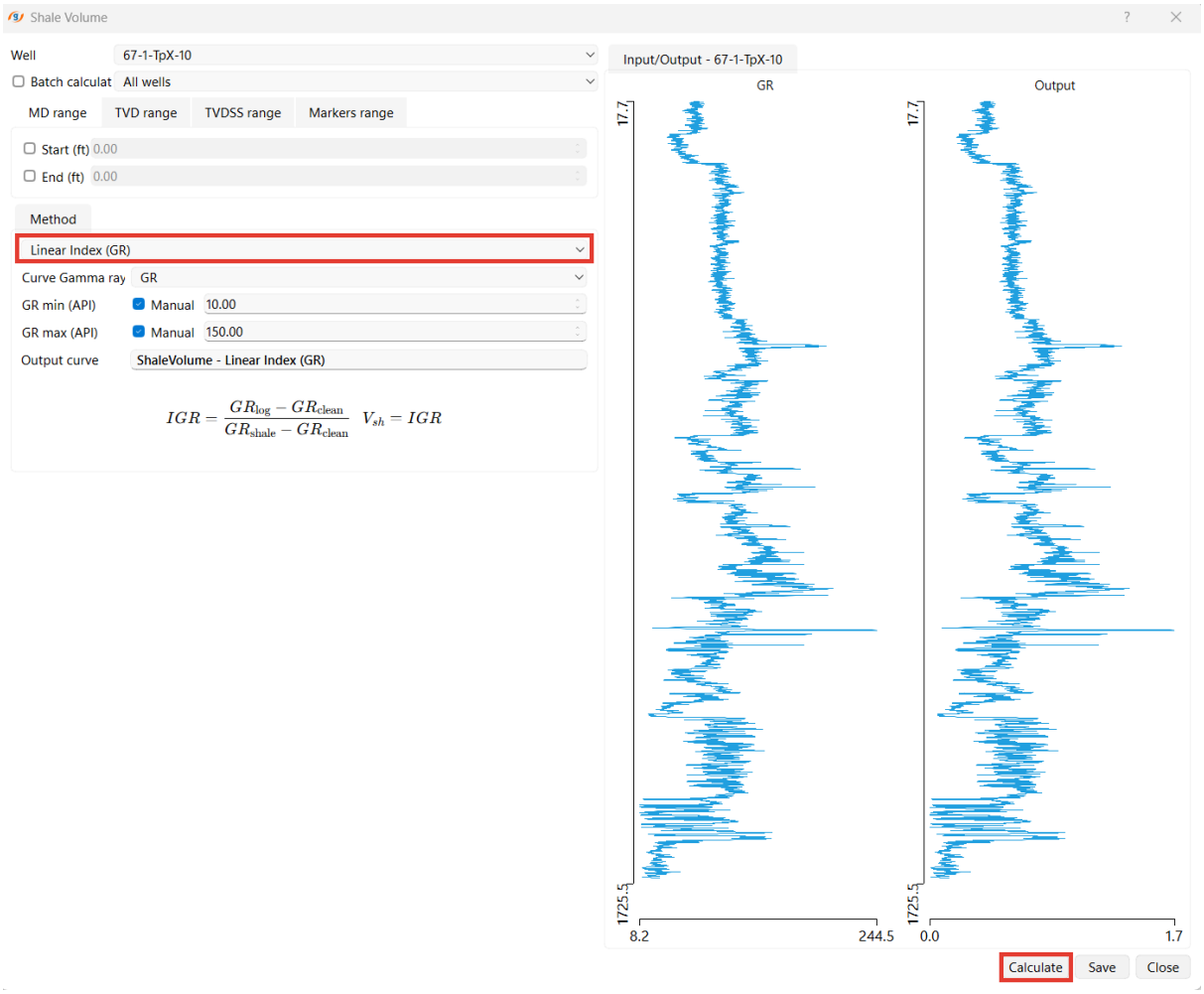
Calculation Methods

1. Linear Index (GR)

This is the simplest approach assuming a linear relationship between Gamma Ray values and shale content.

Use:

- *Quick estimation for clean sand–shale sequences.*
- *Best applied when GR response is approximately linear with shale content.*



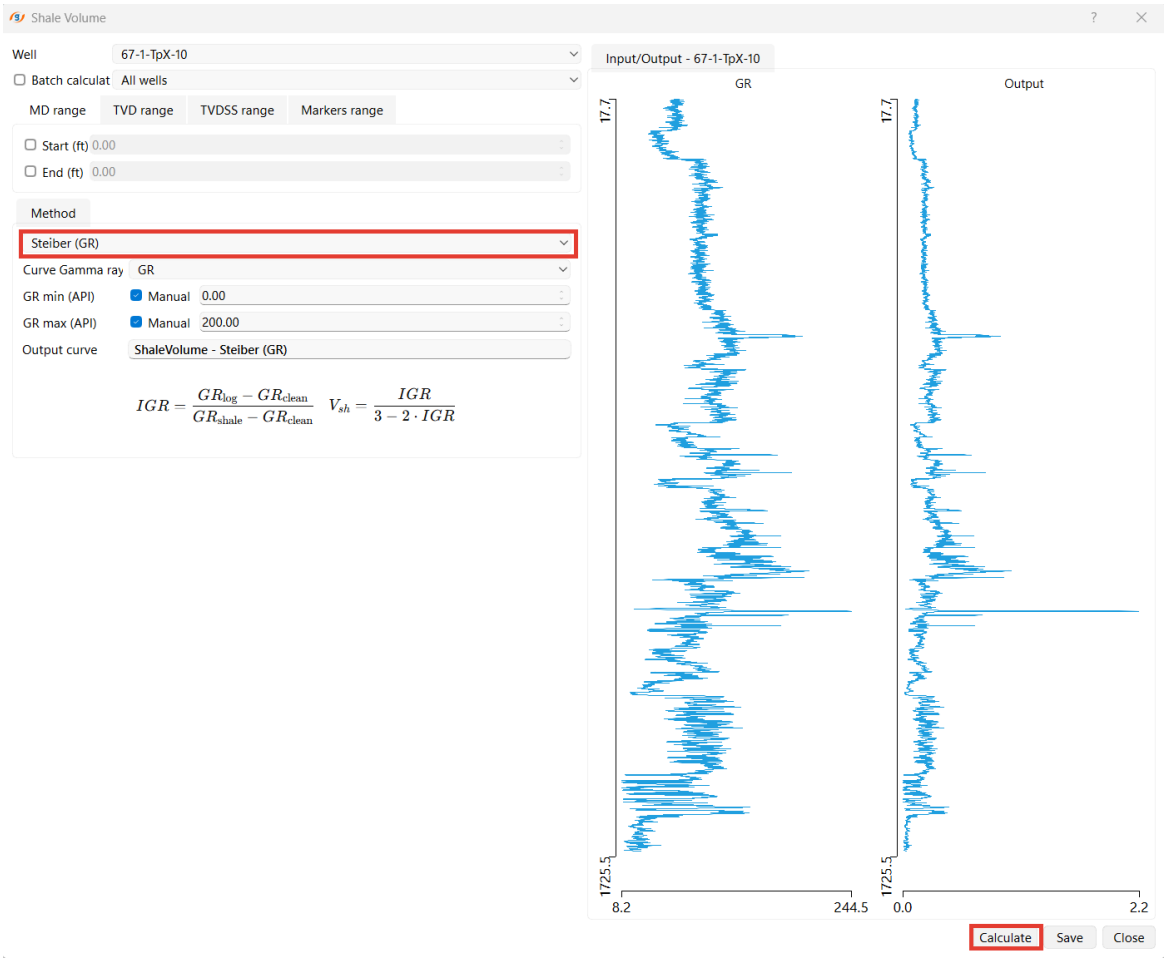
© 2026 Geomage 2003 LTD

2. Steiber (GR)

Introduces a non-linear correction to better reflect compaction and radioactive mineral effects.

Use:

- Suitable for mixed lithologies and compacted formations.
- Reduces overestimation of shale content in clean sands.



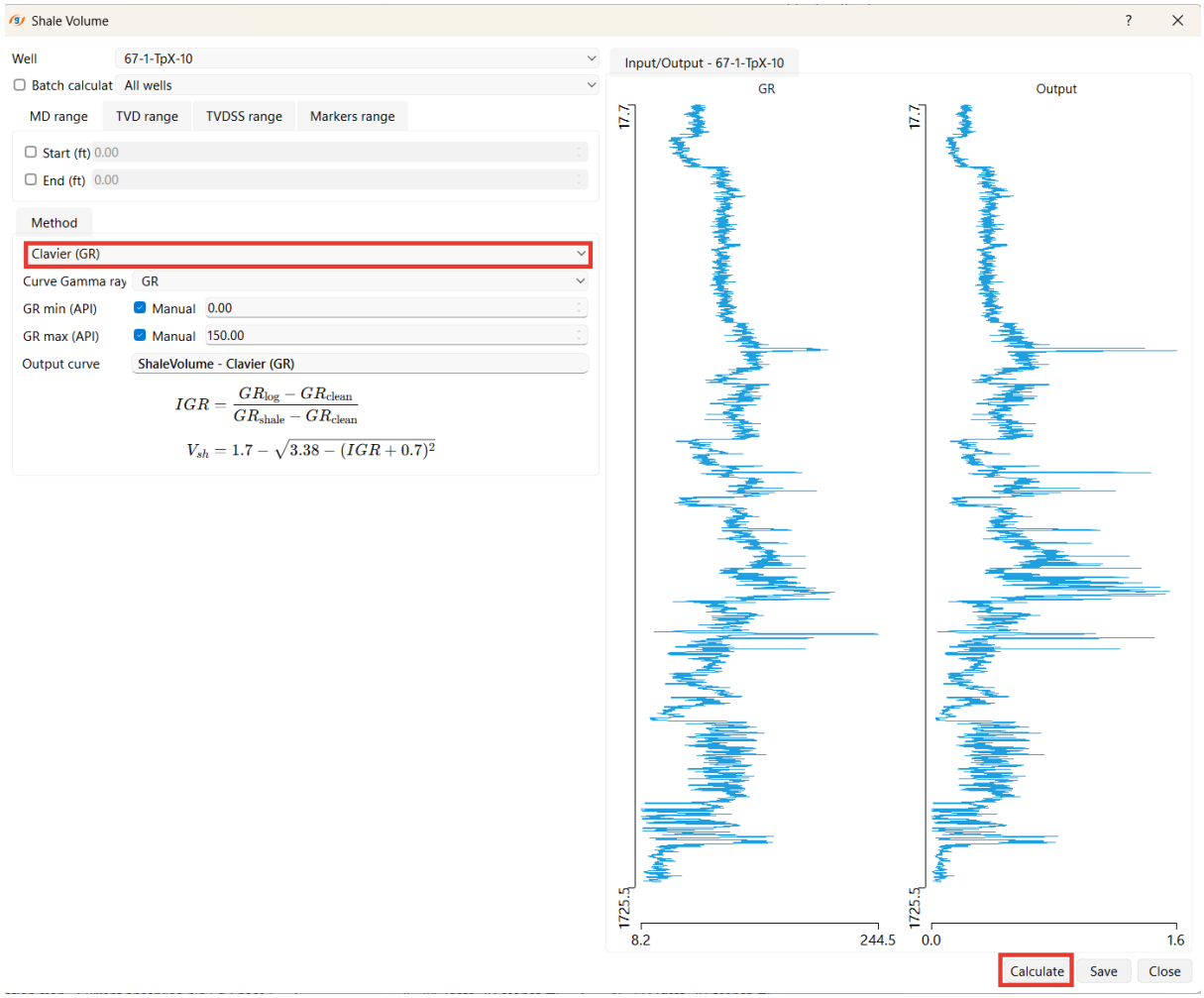
3. Clavier (GR)

An empirical formula that smooths the transition between clean and shaly intervals.

Use:

- *Appropriate for moderately shaly formations.*
- *Often applied to Tertiary clastic reservoirs.*

© 2026 Geomage 2003 LTD



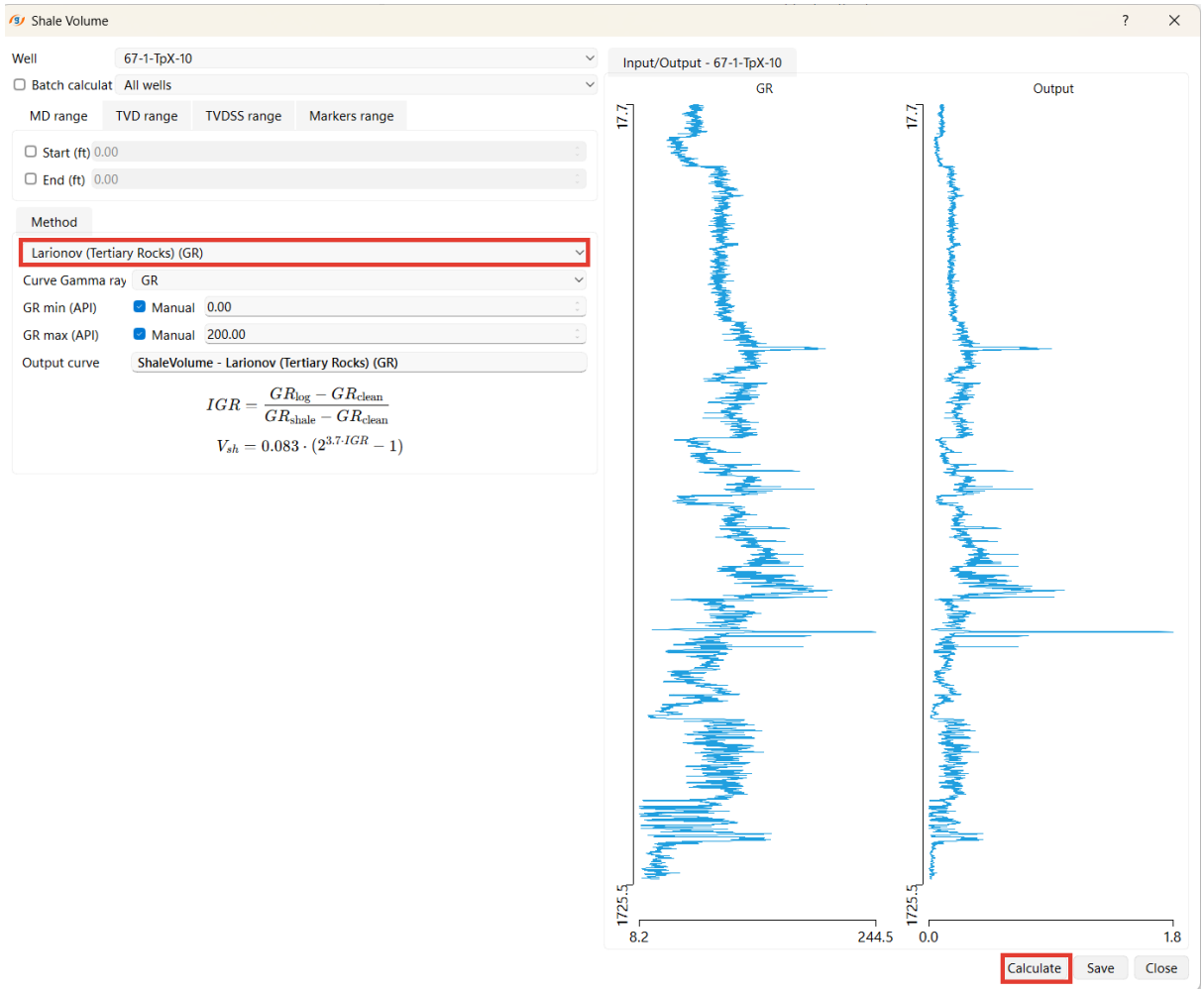
4. Larionov (Tertiary Rocks) (GR)

Developed by Larionov (1969) for young sedimentary sequences with low compaction.

Use:

- *Recommended for Tertiary rocks and soft sediments.*
- *Corrects for overestimated shale content in younger formations.*

© 2026 Geomage 2003 LTD



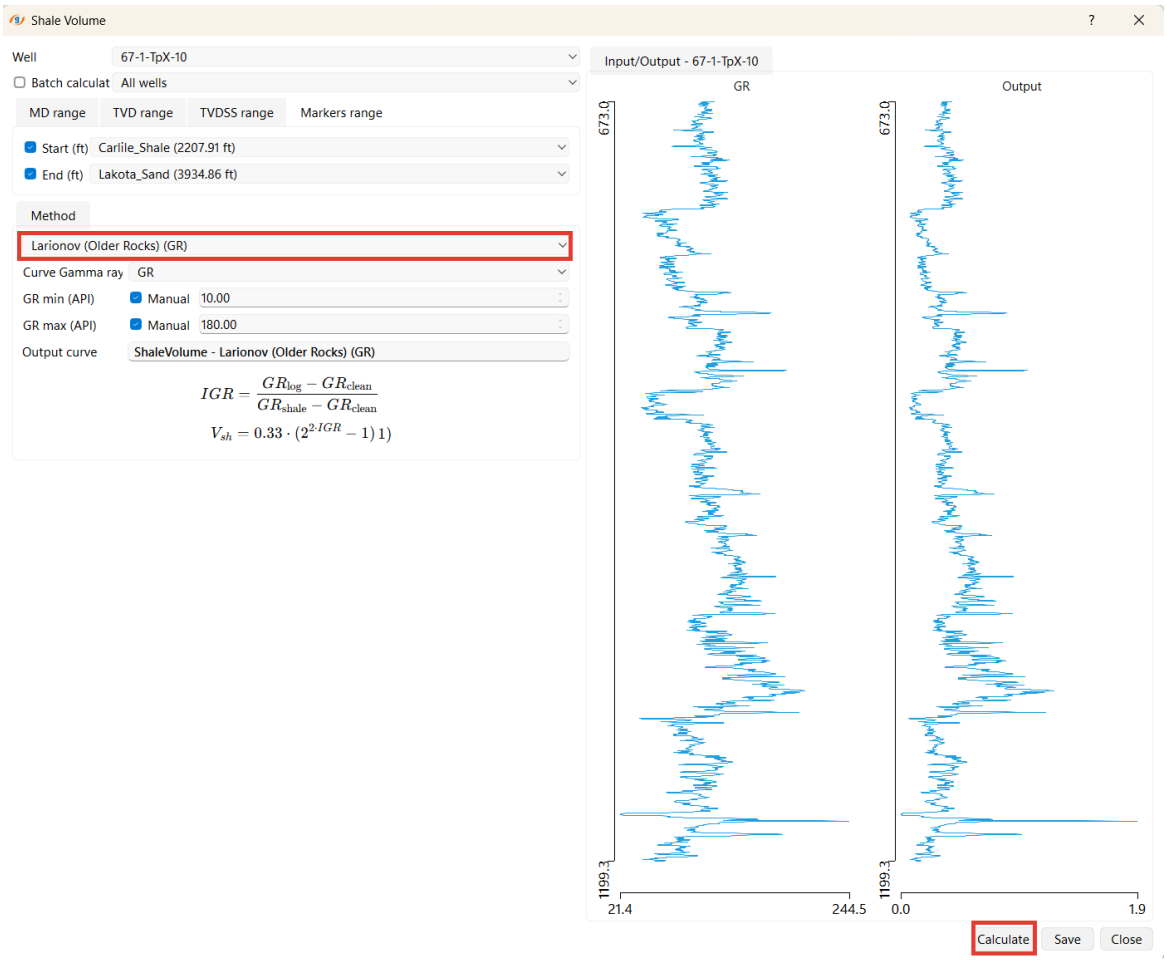
5. Larionov (Older Rocks) (GR)

Modified Larionov model for compacted and older formations (e.g., Paleozoic–Mesozoic).

Use:

- *Suitable for consolidated rocks and older stratigraphic units.*
- *Provides more realistic shale values in deep formations.*

© 2026 Geomage 2003 LTD



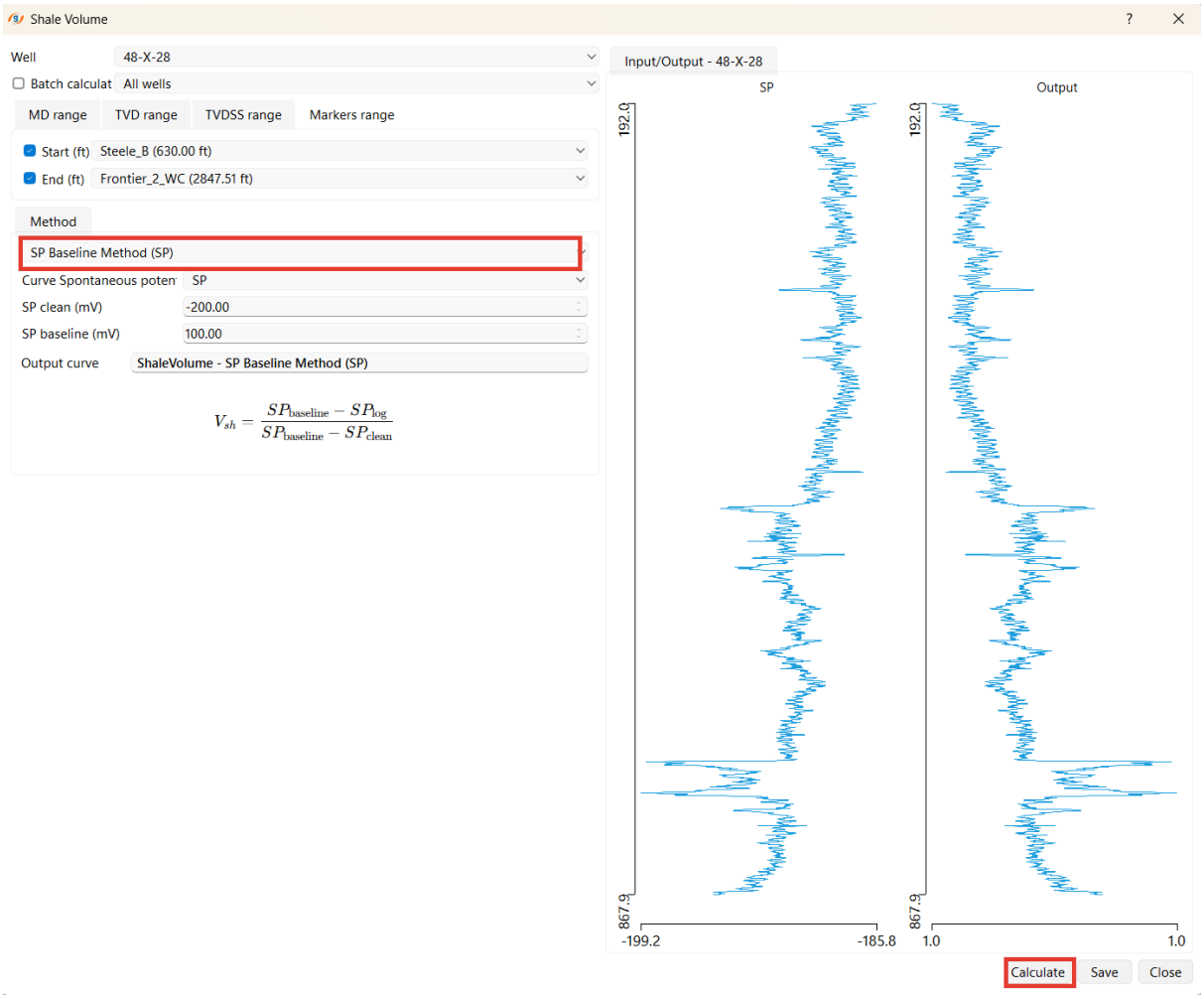
6. SP Baseline Method (SP)

Uses **Spontaneous Potential (SP)** log instead of GR for shale estimation.
The method compares the measured SP curve with reference clean and shale baselines.

Use:

- *Useful when GR data is unavailable or unreliable.*
- *Common in older wells with limited logging suites.*

© 2026 Geomage 2003 LTD



7. Neutron–Density

Estimates shale volume from the separation between the **Neutron Porosity (NPHI)** and **Density (RHOB)** logs, by projecting each sample onto the line joining the clean-sand and shale end points on the neutron–density cross-plot. The result is clipped to the range 0–1.

Input curves: Neutron Porosity and Density.
Parameters (editable):

- **Clean sand:** NPHI clean sand (default 0.10 v/v), RHOB clean sand (default 2.65 g/cc).
- **Shale:** NPHI shale (default 0.35 v/v), RHOB shale (default 2.40 g/cc).

Use:

- Reservoirs where GR is unreliable (e.g. radioactive or feldspathic/mica-rich sands).
- Cross-checking a GR- or SP-based shale volume against a porosity-log estimate.

© 2026 Geomage 2003 LTD

The default output curve name is `ShaleVolume - NeutronDensity`.

Running and saving to Workflow

The wizard has a standard row of buttons along the bottom of the dialog:

- **Save to workflow** – adds the wizard, with all of its current settings (selected wells, input and output curves, method, method parameters, and output names), to the project Workflow 535 as a task that can be re-run later.
- **Run** – executes the calculation with the current settings (this button replaces the former *Calculate* button).
- **Save** – stores the resulting curve in the well.
- **Close** – closes the wizard.

When a saved task is reopened from the Workflow 535 panel, the wizard reopens with every stored setting restored, so it can be reviewed, edited, run, and saved back to the task. In this edit-from-workflow mode the bottom row shows **Save** and **Cancel** instead.

3.1.6.3 Porosity

The **Porosity** wizard calculates total and effective porosity using available well logs such as density, neutron, or a combination of both.
The tool can be applied to a **single well** or in **batch mode**, allowing users to process all wells or selected wells from the **Data Manager**.

Input Parameters

- **Well selection:** Choose one well or enable **Batch calculation** to process multiple wells or a well group from the Data Manager.
- **Depth filter:** The calculation interval can be limited by **MD**, **TVD/TVDSS**, **markers**, or **zones**. On the **Zones range** tab, select one or more zones of a zone set; a selected zone that does not exist in a well is skipped for that well.
- **Curve selection:** Each curve field is named after the log type it expects – **Density**, **Neutron Porosity**, **P-wave** and **Curve Shale volume** – and only the fields required by the selected method are active. Each field is filled automatically with the first curve of that type in the selected well, and the list also offers the other curves of the well. For the **Density** curve, choose kg/m^3, g/cc, or lb/ft^3 from the adjacent unit list.
- **Material properties:** Type the matrix and fluid values used by the selected method, and the shale porosity where it applies. These fields accept any value, and **Matrix density**,

© 2026 Geomage 2003 LTD

Fluid density, **Matrix travel time** and **Fluid travel time** show a tooltip with typical values for common lithologies and fluids when the pointer rests on them. Every value is expressed in the unit shown beside the field, which follows the project measurement system.

- **Output curve:** Define the name of the resulting porosity log. The name can be **edited directly in the wizard** before saving.

Default material properties. The wizard is pre-filled with common presets that can be edited for the local lithology and fluid:

- **Matrix density** – Sandstone, 2.65 g/cc (Limestone 2.71, Dolomite 2.87).
- **Fluid density** – Saline water, 1.05 g/cc.
- **Shale porosity** – 0.2.
- **Matrix travel time** – Sandstone, 180 µs/m (Limestone 155, Dolomite 145).
- **Fluid travel time** – Saline water, 605 µs/m.

The density and travel-time presets listed above are the metric values. The wizard shows each of them converted into the unit displayed beside its field, so a project that uses the imperial measurement system opens with **Matrix density** 165.43 lb/ft³, **Fluid density** 65.55 lb/ft³, **Matrix travel time** 54.86 µs/ft and **Fluid travel time** 184.40 µs/ft.

Calculation Methods

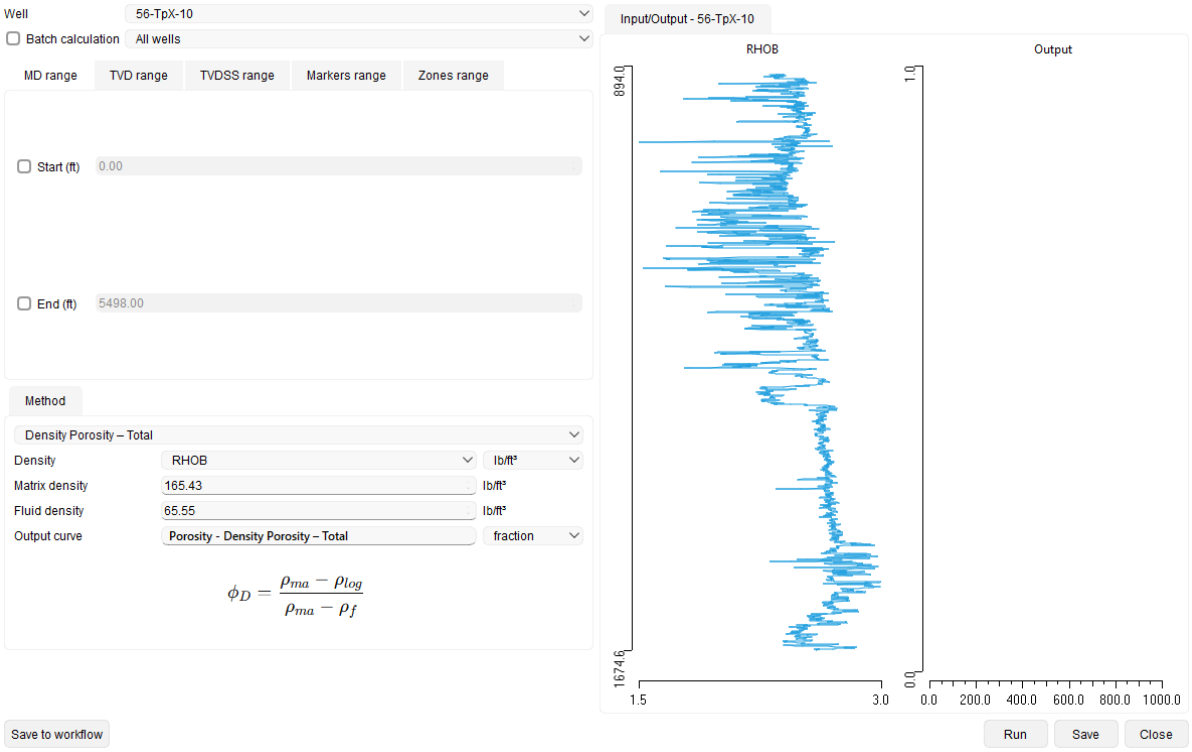
Density Porosity – Total

Calculates porosity directly from the density log assuming a clean (shale-free) formation.

Use:

- For clean formations with minimal shale influence.
- Common for sandstone and carbonate reservoirs.

© 2026 Geomage 2003 LTD



ρ_{ma} – matrix density,

ρ_f – fluid density,

ρ_{log} – measured formation density.

Density Porosity – Effective

Corrects the total porosity for shale content using a shale volume curve and shale porosity.

Use:

- Appropriate for shaly formations.
- Produces an effective porosity excluding bound water in shale.

$\phi D,eff = (\rho ma - \rho log) / (\rho ma - \rho f) - Vsh \cdot \phi sh$

V_{sh} – shale volume,

ϕ_{sh} – shale porosity.

© 2026 Geomage 2003 LTD

Neutron–Density Porosity – Total

Combines the neutron and density porosity responses (cross-plot / average) to give a total porosity that is less sensitive to lithology than either log alone.

Use:

- Mixed or unknown lithology where a single-log estimate is unreliable.
- Requires both Neutron Porosity and Density curves.

Neutron–Density Porosity – Effective

The neutron–density total porosity corrected for shale content using a shale volume curve and the shale porosity, yielding effective porosity.

Use:

- Shaly formations of mixed lithology.
- Requires Neutron Porosity, Density and a shale volume curve.

Sonic Porosity – Total

Calculates total porosity from the sonic (interval transit time) log using the Wyllie time-average relation, with matrix and fluid travel-time constants.

Use:

- Consolidated formations where only a sonic log is available.
- Requires matrix and fluid travel-time values (see defaults above).

Sonic Porosity – Effective

The sonic total porosity corrected for shale content using a shale volume curve and the shale porosity, yielding effective porosity.

Use:

- Shaly, consolidated formations logged with a sonic tool.
- Requires a sonic log and a shale volume curve.

Neutron–Density–Sonic

© 2026 Geomage 2003 LTD

A triple-combo estimate that jointly uses the Neutron, Density and Sonic logs to derive porosity, improving robustness in complex lithology and reducing gas or lithology artefacts present in any single log.

Use:

- Complex or gas-bearing reservoirs where a full logging suite is available.
- Requires Neutron Porosity, Density and Sonic curves.

Running and saving to Workflow

The wizard has a standard row of buttons along the bottom of the dialog:

- **Save to workflow** – adds the wizard, with all of its current settings (selected wells, input and output curves, method, method parameters, material constants, and output names), to the project Workflow 535 as a task that can be re-run later.
- **Run** – executes the calculation with the current settings (this button replaces the former *Calculate* button).
- **Save** – stores the resulting porosity curve in the well.
- **Close** – closes the wizard.

When a saved task is reopened from the Workflow 535 panel, the wizard reopens with every stored setting restored, so it can be reviewed, edited, run, and saved back to the task. In this edit-from-workflow mode the bottom row shows **Save** and **Cancel** instead.

3.1.6.4 Saturation

The **Saturation** wizard estimates water saturation from a deep-reading resistivity log and a porosity log using industry-standard equations (Archie, Simandoux, and their shale-corrected variants). The result is a water-saturation curve written on the scale of the selected output unit (**fraction** by default), suitable for net-pay and reserves calculations.

The tool can be applied to a **single well** or in **batch mode**, allowing users to process all wells or selected wells from the **Data Manager**.

Launching the wizard

Open the **Petrophysics** ribbon bar and click **Saturation**. Choose the target well, select a method, supply the required input curves and constants, and preview the result in the chart before saving.

© 2026 Geomage 2003 LTD

Input Parameters

- **Well selection:** Choose one well or enable **Batch calculation** to process multiple wells or a well group.
- **Depth filter:** The calculation interval can be limited by **MD**, **TVD/TVDSS**, **markers**, or **zones**. On the **Zones range** tab, select one or more zones of a zone set; a selected zone that does not exist in a well is skipped for that well.
- **Method:** Select one of the equations described below. The parameter grid adapts automatically to the chosen method: the curves and constants that the method does not use are hidden.
- **Input curves:** True resistivity *Rt* (required), and depending on the method one of *Total porosity* or *Effective porosity*; plus shale volume *Vsh* for the shaly variants. **Curve Total porosity**, **Curve Effective porosity** and **Curve Shale volume** each have their own unit list; set it to **fraction** or **%** to match how the log is stored. **Curve Resistivity** has no unit list: it shows a fixed **ohm×m** unit label.
- **Formation constants:** Tortuosity factor *a* (default 1), cementation exponent *m* (default 2), saturation exponent *n* (default 2), formation water resistivity *Rw* (default 0.1 ohm·m), shale resistivity *Rsh* (default 3 ohm·m), and shale-correction exponent *x* (default 1.5, Indonesia method only). All constants are editable.
- **Output curve:** Name of the resulting water-saturation log. The name is filled in automatically from the selected method, in the form **Saturation - <method name>**, and it is refilled every time the method is changed. The name can also be **edited directly in the wizard** before saving; because a change of method replaces a name typed by hand, enter a custom name last. The row also has its own unit list, which sets the scale of the result: **fraction** (the default) writes the water-saturation values on the 0–1 scale, while **%** writes them in percent and stores **%** as the unit of the saved curve.

Calculation Methods

Pick the method that matches the lithology and the logs available. The Method panel displays the formula image and exposes only the parameters consumed by the selection.

Archie (Total porosity saturation)

The classical Archie equation applied to the total porosity curve. Assumes a clean, water-wet formation where conductivity is controlled by the brine in the pore network.

Inputs: *Rt*, total porosity, *Rw*, *a*, *m*, *n*.

© 2026 Geomage 2003 LTD

Use: clean sandstones and carbonates with negligible clay content.

Simandoux (Effective porosity saturation)

Extends Archie with an additive conductive term that represents shale conductivity, applied to the effective porosity curve. Reduces the overestimation of water saturation in shaly sands that Archie produces.

Inputs: *Rt*, effective porosity, *Rw*, *a*, *m*, *n*, shale volume *Vsh*, shale resistivity *Rsh*.
Use: shaly reservoirs with dispersed clay where a quantitative Vsh curve is available.

Indonesia (Effective porosity saturation)

The Poupon–Leveaux "Indonesia" shaly-sand model, applied to the effective porosity curve. It combines the clean-sand and shale conductivity contributions through a *Vsh*-weighted term, which makes it more stable than Simandoux at high shale volumes. This is the only method that also uses the shale-correction exponent *x*.

Inputs: *Rt*, effective porosity, *Rw*, *a*, *m*, *n*, shale volume *Vsh*, shale resistivity *Rsh*, shale-correction exponent *x*.
Use: laminated or dispersed shaly sands, particularly where formation water is fresh and Simandoux becomes unstable.

Total shale (Total porosity saturation)

A total-shale model that adds a shale-conductivity term proportional to *Vsh* to the Archie response, evaluated on the total porosity curve.

Inputs: *Rt*, total porosity, *Rw*, *a*, *m*, *n*, shale volume *Vsh*, shale resistivity *Rsh*.
Use: shaly formations where a total-porosity reference is preferred and a Vsh curve is available.

Total shale (Effective porosity saturation)

The same total-shale model evaluated on the effective porosity curve, so the resulting saturation is referenced to the effective (non-clay-bound) pore space.

Inputs: *Rt*, effective porosity, *Rw*, *a*, *m*, *n*, shale volume *Vsh*, shale resistivity *Rsh*.

© 2026 Geomage 2003 LTD

Use: shaly reservoirs where effective porosity is preferred for reserves accounting.

Dual water (Total porosity saturation)

A dual-water model that distinguishes bound water associated with clay from free formation water, applied to the total porosity curve. It accounts for the extra conductivity of clay-bound water and gives more realistic saturations in clay-rich intervals.

Inputs: *Rt*, total porosity, *Rw*, *a*, *m*, *n*, shale volume *Vsh*, shale resistivity *Rsh*.
Use: clay-rich reservoirs where bound-water conductivity is significant and the total-porosity system is preferred.

Running and saving to Workflow

The wizard has a standard row of buttons along the bottom of the dialog:

- **Save to workflow** — adds the wizard, together with all of its current settings (selected wells, input and output curves, method, method parameters, user-defined constants, and output names), to the project Workflow 535 as a task that can be re-run later.
- **Run** — executes the calculation with the current settings (this button replaces the former *Calculate* button).
- **Save** — stores the resulting curve in the well.
- **Close** — closes the wizard.

When a saved task is reopened from the Workflow 535 panel, the wizard reopens with every stored setting restored, so it can be reviewed, edited, run, and saved back to the task. In this edit-from-workflow mode the bottom row shows **Save** and **Cancel** instead.

Results

The wizard produces a single water-saturation curve per processed well. The value returned by the selected equation is stored as it is, on the scale of the chosen output unit: **fraction** (the default) writes it on the 0–1 scale, and **%** on the 0–100 scale. No lower or upper limit is applied, so unsuitable input curves or constants can produce values outside that scale. A preview chart plots the result against the input resistivity and porosity logs, so check it for non-physical excursions before saving. Saved curves can be displayed in the **Well log view** and consumed by downstream tools such as **Volumetric calculations**.

© 2026 Geomage 2003 LTD

Recommendations

- Calibrate *a*, *m*, and *n* against core-derived Sw or capillary-pressure data where available; default Archie values (a = 1, m = n = 2) rarely fit real rocks precisely.
- Use the **Shale volume** tool first to produce a reliable *Vsh* curve before switching to any of the shaly-sand methods (Simandoux, Indonesia, Total shale, or Dual water).
- For batch mode, ensure that every well in the group carries the required input curves. A well whose required curve is missing is skipped, and the wizard names that well and the missing input in the message shown when the run finishes.

See Also

- Petrophysics 49
- Porosity 65
- Shale Volume 58
- Lithology 77
- Volumetric calculations 790

3.1.6.5 Pore pressure

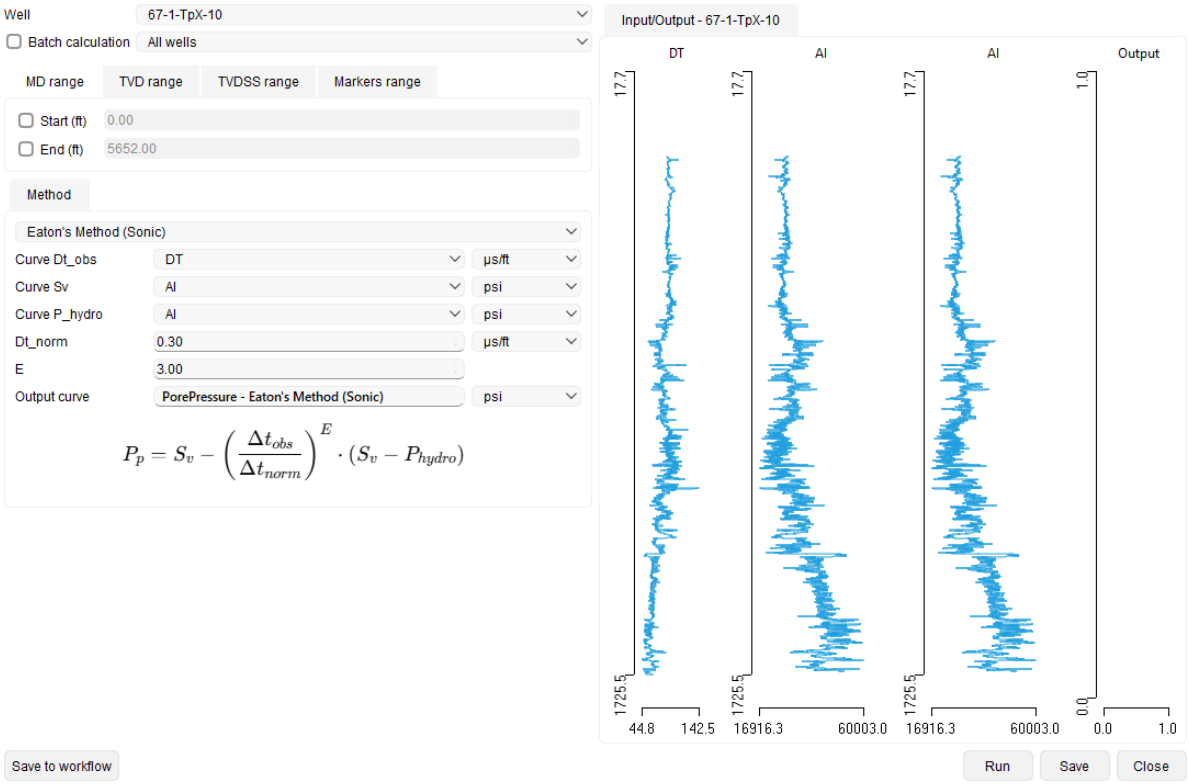
The **Pore pressure** wizard estimates formation pore pressure from well log data using standard prediction methods such as Eaton's and Bowers'. The result is a pore-pressure curve that can be used for drilling planning, geomechanical modelling, and seal-integrity analysis.

The tool can be applied to a **single well** or in **batch mode**, allowing users to process all wells or selected wells from the **Data Manager**.

Launching the wizard

Open the **Petrophysics** ribbon bar and click **Pore pressure**. Select the target well, pick a prediction method, and provide the required input curves and calibration parameters.

© 2026 Geomage 2003 LTD



Input Parameters

- **Well selection:** Choose one well or enable **Batch calculation** to process multiple wells or a well group.
- **Depth filter:** The calculation interval can be limited by **MD**, **TVD/TVDSS**, or **markers**.
- **Method:** Choose one of the prediction methods described below. The parameter set displayed in the **Method** group adapts to the selection, and only the curves and constants required by the chosen method remain editable.
- **Input curves:** Depending on the method the wizard requires one or several of: sonic slowness (*Dt_obs*), interval velocity (*Vint*), observed resistivity (*Robs*), normal-compaction resistivity (*Rnorm*), overburden stress (*Sv*), hydrostatic pressure (*P_hydro*), porosity, and effective stress (*Sigma_eff*).
- **Calibration constants:** Method-specific parameters such as *Dt_norm*, Eaton exponent *E*, Bowers coefficients *A* and *B*, compaction parameters ϕ_0 and β, and the unloading exponent.
- **Output curve:** Name and unit of the resulting pore-pressure log (MPa or psi). The name can be **edited directly in the wizard** before saving.

Calculation Methods

© 2026 Geomage 2003 LTD

Pick the method that matches the logs available in the well. The wizard shows the formula of the selected method below its parameter fields in the **Method** group.

Units — metric and imperial

Every numeric field carries a unit selector that follows the project measurement system (**Metric** or **Imperial**). The wizard converts the inputs internally before calculating:

- Sonic slowness (*Δt*): **µs/m** (metric) / **µs/ft** (imperial).
- Interval velocity (*Vint*): **m/s** / **ft/s**.
- Stresses and pressures (*Sv*, *Pp*, *P_hydro*, *Sigma_eff*): **MPa** / **psi**.
- Compaction coefficient (*β*): **MPa^{-1}** / **psi^{-1}**.
- Resistivity (*Robs*, *Rnorm*): **ohm·m** in both systems.
- Porosity (*ϕ*): **fraction** in both systems.

Eaton's Method (Sonic)

Predicts pore pressure from the ratio of the observed sonic slowness to the normal-compaction trend, scaled by the difference between overburden stress and hydrostatic pressure.
Inputs: sonic curve *Dt_obs*, normal-compaction value *Dt_norm*, overburden stress curve *Sv*, hydrostatic pressure curve *P_hydro*, Eaton exponent *E*.
Use: shaly sections where a normal-compaction trend on the sonic log can be established.

Resistivity Ratio Method (Traugott Equation)

Driven by resistivity logs. Pore pressure rises where the measured resistivity deviates below the normal-compaction trend; the departure is raised to an empirical exponent (Traugott) and applied to the difference between overburden stress and hydrostatic pressure.
Inputs: observed resistivity *Robs*, normal-compaction resistivity *Rnorm*, overburden stress *Sv*, hydrostatic pressure *P_hydro*, ratio exponent.
Use: wells with good resistivity logs in shales, where no reliable sonic measurement is available.

Bowers' Method

Relates interval velocity to effective stress via an empirical loading curve. The method supports both loading and unloading regimes and therefore handles undercompaction and unloading-induced overpressure.

© 2026 Geomage 2003 LTD

Inputs: interval velocity curve *Vint* (or sonic), overburden stress *Sv*, Bowers coefficients *A* and *B*, unloading exponent.
Use: areas where unloading mechanisms (fluid expansion, hydrocarbon generation) drive overpressure and Eaton's assumption of continuous loading breaks down.

Sonic Porosity Method (Effective Stress)

Derives effective stress from a porosity-versus-depth compaction trend $\phi(z) = \phi_0 \cdot \exp(-\beta \cdot z)$, then subtracts it from the overburden stress to obtain pore pressure. The porosity curve is typically derived from the sonic log.
Inputs: porosity curve, overburden stress *Sv*, compaction parameters ϕ_0 and β.
Use: wells with reliable porosity logs and well-constrained compaction trends.

Overburden – Effective Stress Method

Uses an existing *Sigma_eff* curve (for example produced by a geomechanical workflow) and subtracts it from the overburden stress. Useful for importing results from other software or coupling with rock-physics modelling inside g-Space.

Running and saving to Workflow

The wizard has a standard row of buttons along the bottom of the dialog:

- **Save to workflow** — adds the wizard, together with all of its current settings (selected wells, input and output curves, method, method parameters, user-defined constants, and output names), to the project Workflow 535 as a task that can be re-run later.
- **Run** — executes the calculation with the current settings (this button replaces the former *Calculate* button).
- **Save** — stores the resulting curve in the well.
- **Close** — closes the wizard.

When a saved task is reopened from the Workflow 535 panel, the wizard reopens with every stored setting restored, so it can be reviewed, edited, run, and saved back to the task. In this edit-from-workflow mode the bottom row shows **Save** and **Cancel** instead.

Results

The wizard produces a single pore-pressure curve per processed well. The **Input/Output** panel plots the curves selected as inputs next to the resulting **Output** track, so calibration issues can

© 2026 Geomage 2003 LTD

be caught before saving. Saved curves are stored in the well's LAS files and can be viewed in the **Well log view** or exported through the **Export data** menu.

Recommendations

- Calibrate the chosen method against direct pressure measurements (RFT, MDT, mud weight) in offset wells before running it in blind mode.
- Use the Base properties wizard beforehand to prepare consistent Hydrostatic pressure and Overburden stress curves — most methods rely on them.
- For batch mode, verify that every well in the group carries the input curves expected by the chosen method; missing inputs will leave the output curve empty for those wells.

See Also

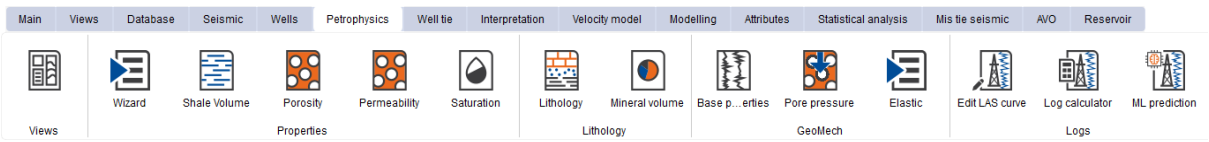
- Petrophysics 49
- Base properties 55
- Porosity 65
- Saturation 69
- Lithology 77

3.1.6.6 Lithology

The **Lithology interpretation** tool assigns a lithotype code to each sample in the selected interval of a well by comparing the input well logs against user-defined value ranges. The result is a single lithology curve where each sample carries the code of the dominant lithotype (for example sand, shale, limestone).

The tool can be applied to a **single well** or in **batch mode**, allowing users to process all wells or selected wells from the **Data Manager**.

Open the Petrophysics 49 ribbon bar and click **Lithology** in the **Lithology** group. Select the target well (or enable **Batch calculation** to process a well group), then define lithotypes and run the calculation.

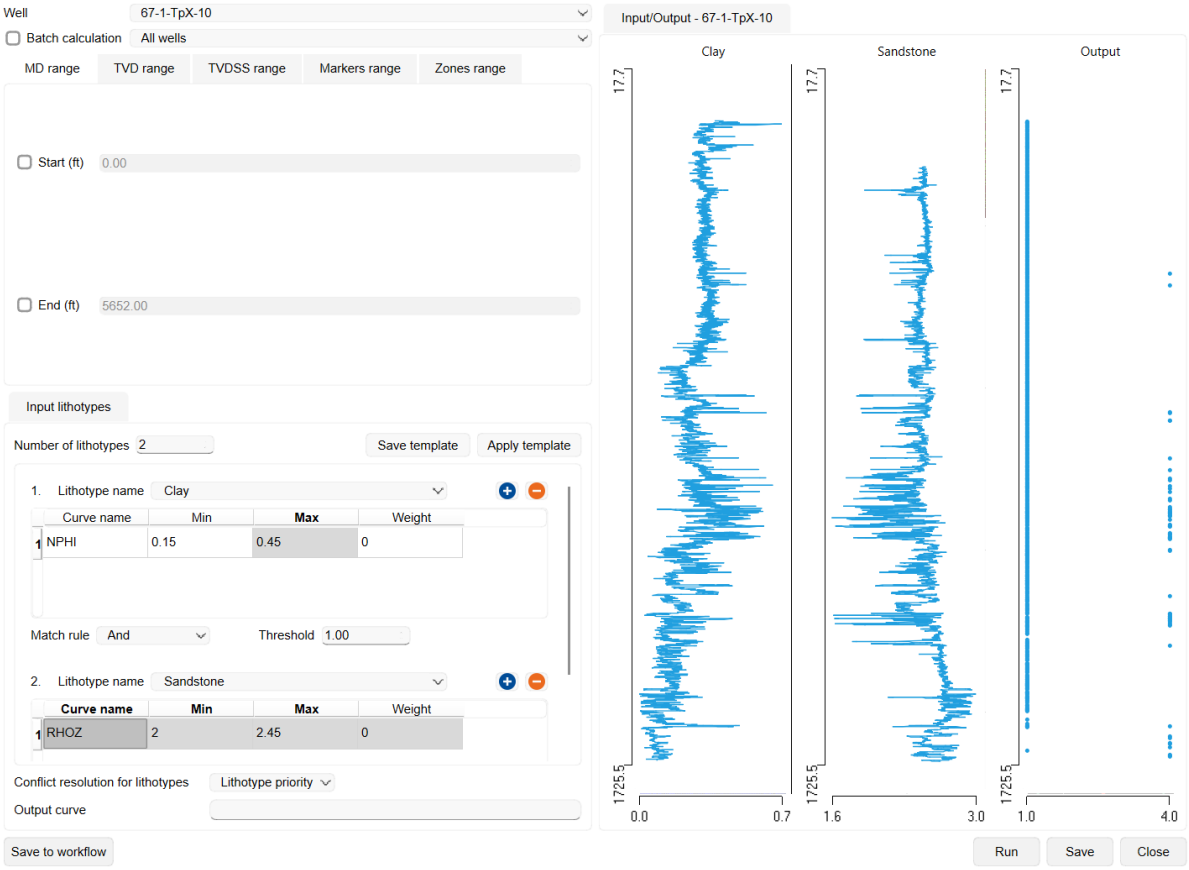


Input Parameters

© 2026 Geomage 2003 LTD

- **Well selection:** Choose one well or enable **Batch calculation** to process multiple wells or a well group from the Data Manager.
- **Depth filter:** Define the interpretation interval using **MD range**, **TVD range**, **TVDSS range**, **Markers range** or **Zones range**. On the Zones range tab, choose one or more zones from a zone set. If more than one eligible zone set is available, choose it from **Zone set**; use **Search zone** to filter the list and **Select all** to select or clear the visible rows. The selection applies only while this tab is active. The selected lithotypes, thresholds, match rules and conflict resolution are applied inside the union of the valid selected intervals; output stays undefined elsewhere, and the preview shades each valid interval in its zone color. In batch mode, boundaries are resolved separately for each well. A missing marker dataset or boundary, an inverted interval, or an interval without log samples is skipped for that well and named in the report while other valid zones and wells continue.
- **Number of lithotypes:** Defines how many lithotype rows appear in the Input lithotypes list.
- **Lithotype name:** Select the lithotype label from the predefined lithology map.
- **Input curves:** All available curves from the current project can be selected for lithology identification.
- **Curve thresholds:** Define minimum and/or maximum values for each selected curve.
- **Match rule:** Defines how input curves are evaluated for a lithotype.
- **Threshold:** Used only in Weight mode. Defines the minimum accumulated score required to assign the lithotype.
- **Conflict resolution for lithotypes:** Defines how overlapping lithotype matches are handled.
- **Output curve:** Name of the resulting lithology curve. Default is Lithology.

© 2026 Geomage 2003 LTD



Templates and history

The wizard stores previously used lithotype definitions so they can be reapplied without re-entry:

- **Save template** — stores the current lithotype list under a user-supplied name for reuse in other projects or wells.
- **Apply template** — loads a saved template or a recent entry from the wizard's history into the current session.

Running and saving to Workflow

The wizard has a standard row of buttons along the bottom of the dialog:

- **Save to workflow** — adds the wizard, together with all of its current settings (selected well or batch selection, active depth filter, selected zone set and zones, lithotype definitions with their input curves, thresholds and match rules, conflict resolution mode and output curve name), to the project Workflow 535 as a task that can be re-run later.
- **Run** — performs the interpretation with the current settings and updates the preview (this button replaces the former *Calculate* button).

© 2026 Geomage 2003 LTD

- **Save** — stores the resulting lithology curve in the well.
- **Close** — closes the wizard.

When a saved task is reopened from the Workflow 535 panel, the wizard restores its stored settings so they can be reviewed, edited and saved back to the task. For a task saved from **Zones range**, the tab opens automatically when at least one saved zone is still available, with all available saved zones selected; unavailable zones are removed from the setup and reported. In this edit-from-workflow mode the bottom row shows **Save** and **Cancel** instead.

Match Rules

AND

All selected curves must satisfy their conditions for the lithotype to be assigned.

Algorithm

1. Evaluate all selected curves.
2. Compare curve values against Min/Max thresholds.
3. If all curves match:
   - assign lithotype.
4. If at least one curve fails:
   - reject lithotype.

OR

At least one selected curve must satisfy its condition for the lithotype to be assigned.

Algorithm

1. Evaluate all selected curves.
2. If any curve matches:
   - assign lithotype.
3. If no curves match:
   - reject lithotype.

Method Priority

Curves are evaluated in user-defined order. The first matching curve defines the lithotype.

Algorithm

1. Evaluate curves from top to bottom.
2. If the current curve matches:
   - assign lithotype immediately.

© 2026 Geomage 2003 LTD

3. If the curve fails:
   - continue with the next curve.
4. Missing curves are skipped.
5. If no curves match:
   - reject lithotype.

Curve availability behavior

Curve availability	Behavior
All curves available	Evaluate in order
Some curves missing	Skip missing curves
All curves missing	Lithotype becomes Undefined

Weight

Each curve contributes a partial score toward lithotype identification.

Core conditions

- All weights must sum to 1.0.
- Only matched curves contribute their weights.
- The final score is compared against the Threshold value.

Algorithm

1. Evaluate all selected curves.
2. Add weights of matched curves.
3. Calculate the total score.
4. Compare the score against Threshold.
5. If score ≥ Threshold:
   - assign lithotype.
6. If score < Threshold:
   - reject lithotype.

Curve	Weight	Matched
GR	0.2	Yes
Vshale	0.3	No
Porosity	0.5	Yes

Conflict Resolution for Lithotypes

When several lithotypes match the same depth sample, the wizard resolves the conflict using one of the following modes.

© 2026 Geomage 2003 LTD

Lithotype priority

The lithotype higher in the lithotype list has higher priority.

Algorithm

1. Evaluate all lithotypes.
2. If multiple lithotypes match:
   - assign the lithotype with highest priority.

Undefined

If multiple lithotypes match:

- the sample is written as Undefined.

This mode is useful for manual review and interpretation refinement.

Displaying lithology in the Well Correlation view

In the Well correlation view 296, right-click the curve header, open **Curve Settings** and set the **Curve Type** to **Lithology**. The curve menu then offers a **Lithology Map** action for assigning a color and fill pattern to each lithology class (for example clay, shale, sandstone, limestone, dolomite). See Working with Palettes 323 for details.

See Also

- Petrophysics 49
- Porosity 65
- Shale Volume 58
- Saturation 69
- Data manager 124
- Well correlation view 296
- Working with Palettes 323

3.1.6.7 ML Log Prediction

ML Log Prediction is a machine-learning wizard that trains a model on existing well logs and then uses it to predict a missing or target log across other wells. The model is a gradient-boosted decision tree (CatBoost), and several models can be created, evaluated, and stored within a single project.

© 2026 Geomage 2003 LTD

The wizard is organised into three tabs — **Training**, **Analysis**, and **Calculation** — that follow the natural workflow: build a model, check its quality, then apply it to generate predicted curves.

Purpose

The tool is designed for:
- Filling gaps where a log is missing in some wells but present in others
- Reconstructing a target curve from other available logs
- Building reusable prediction models that can be applied to all wells in a project
- Quality-controlling predictions through feature-impact and cross-correlation analysis

Requirements

- Wells with well logs loaded into the project and available in the **Data Manager**
- One or more input curves (predictors) and a target curve with valid samples in the selected training well or batch
- Matching curve names or mnemonics across wells so that the selected predictors and target can be found
- The **CatBoost** training engine. If it is not installed when the wizard opens, g-Space asks whether to download and install it; internet access is required for that download. If installation is not completed, the wizard closes.

Launching the wizard

Open the Petrophysics 49 ribbon bar and click **ML Log prediction**. The wizard opens as a dialog window titled **ML Log Prediction** with three tabs: **Training**, **Analysis**, and **Calculation**.

Step-by-Step Guide

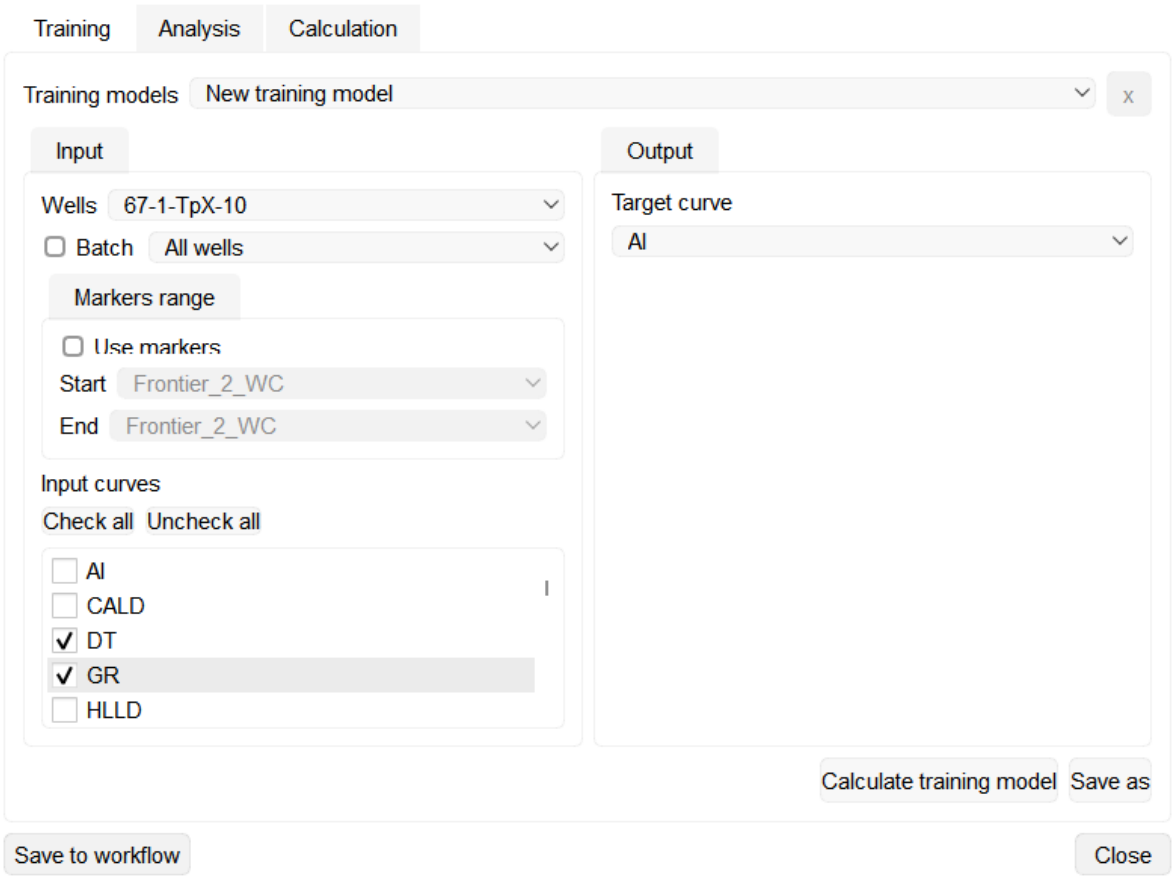
© 2026 Geomage 2003 LTD

Work through the tabs in order: first train and save a model on the Training tab, review its quality on the Analysis tab, then apply it to wells on the Calculation tab.

Training Tab

The Training tab is used to create and train a model. Multiple models can be created and stored within a single project.

Training models — at the top of the tab, the **Training models** dropdown lists every model saved in the project. Its default entry is **New training model**. Select an entry to switch between models, or click the **x** button to delete the currently selected model (a confirmation prompt is shown).



Input group — defines the training dataset:

- **Wells** — select one well. To train on multiple wells, enable **Batch** and choose **All wells** or an available filter, folder, or well section from the adjacent list.

© 2026 Geomage 2003 LTD

- **Markers range** — enable **Use markers** to restrict training to the MD interval between the selected **Start** and **End** markers. When **Use markers** is cleared, the full log interval is used and the marker controls are disabled.
- **Input curves** — the checkable list of logs used as predictors. Use **Check all** or **Uncheck all** to toggle the whole list. At least one input curve is required.

Output group — set the **Target curve**: the single log the model will learn to predict. A target curve is mandatory for training.

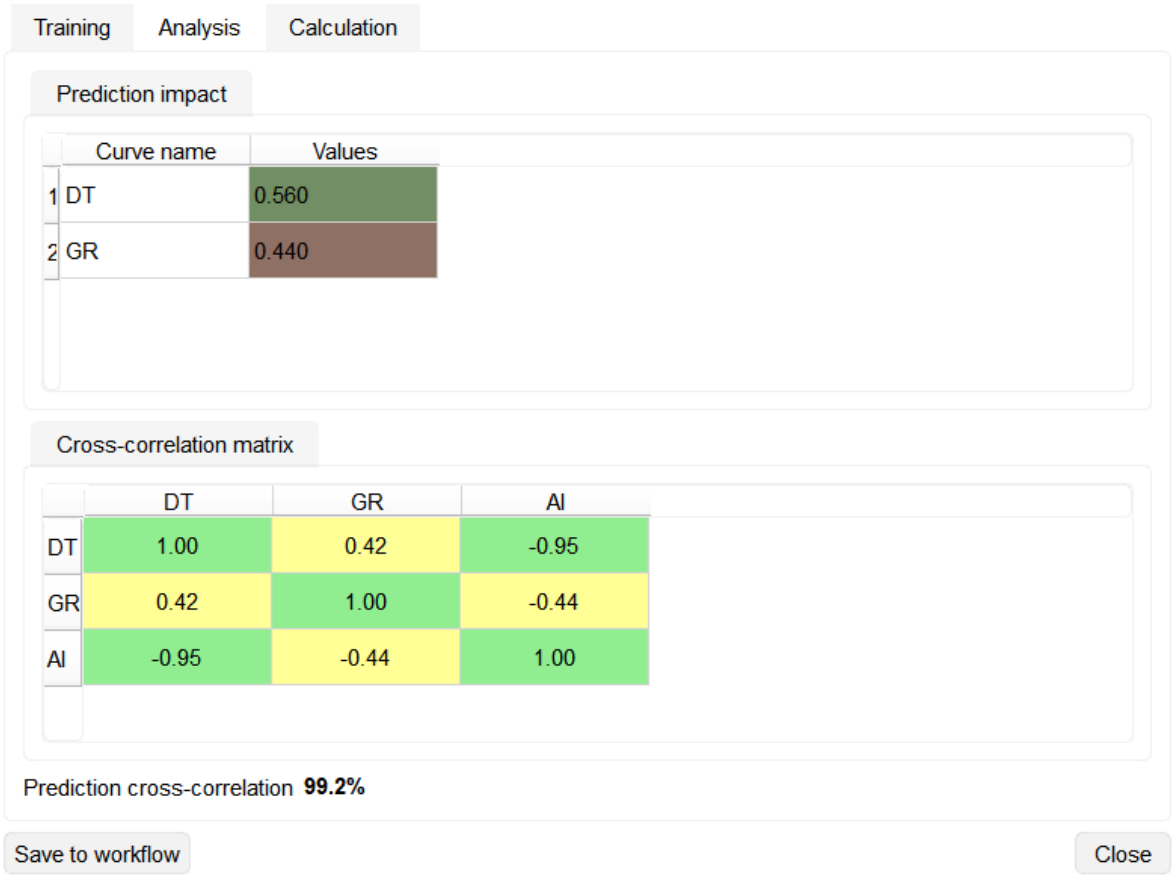
Training and saving the model:

1. Click **Calculate training model** to train the model. On completion a message reports the model's *MAE*, *RMSE*, and prediction correlation, and the **Analysis** tab is populated with the results.
2. Click **Save as** and enter a model name when prompted. The model must be saved before it can be used on the **Calculation** tab; saved models appear in the **Training models** dropdown and are stored with the project.

Analysis Tab

The Analysis tab provides model evaluation and quality control. It reflects the currently trained or selected model.

© 2026 Geomage 2003 LTD



- **Prediction impact** — a table (columns **Curve name** and **Values**) showing how much each input curve contributes to the prediction, with a colour bar for quick visual comparison. Use it to identify the key predictors and to detect weak or redundant inputs.
- **Cross-correlation matrix** — the correlation between the input curves and the output curve, colour-coded by absolute correlation strength: green for strong, yellow for medium, and red for weak correlations.
- **Prediction cross-correlation** — a single performance indicator (for example **98.0%**) expressing the agreement between predicted and actual values. It reads **N/A** until a model has been trained.

Calculation Tab

The Calculation tab applies a saved model to generate predicted logs.

© 2026 Geomage 2003 LTD

1. Choose a model from the **Training model** dropdown. Only saved models are listed.
2. Select one **Wells** entry, or enable **Batch** and choose **All wells** or an available filter, folder, or well section.
3. Set the **Output curve** name. The default is `ML Curve 1`, and it can be edited.
4. To save the configured prediction as a job instead of running it now, click **Save to workflow**; g-Space adds it to the current workflow 535. When you reopen that job for editing, **Save** updates it and **Cancel** closes without updating it.
5. Click **Run** to generate the predicted curves. A new log is written to each processed well and added to the **Data Manager**; a message reports how many wells were processed and how many, if any, failed.

Results

For every successfully processed well the wizard creates a new well log holding the model's prediction along the well's depth. The curve is saved to the **Data Manager** and can be displayed in the **Well log view** alongside the original curves. The completion message reports any wells that could not be processed.

© 2026 Geomage 2003 LTD

Recommendations

- Choose input curves that are physically related to the target log, then use the **Prediction impact** table to drop weak or redundant predictors and re-train.
- Train only on wells that carry both the predictors and the target curve, and use a marker interval to focus the model on the reservoir section.
- Aim for a high **Prediction cross-correlation** and low *MAE* / *RMSE* before saving a model for production use.
- Create several models (for example per formation or per log type) and keep them in the project; the Analysis tab always reflects the currently selected model.
- Re-train and re-save a model whenever you change its input curves, target curve, or training wells.

See Also

- Petrophysics 49
- Porosity 65
- Saturation 69
- Shale Volume 58
- Well log calculator 558
- Workflows 535

3.1.6.8 Mineral Volume Estimation

Mineral Volume Estimation is a petrophysical wizard that estimates mineral (lithology) volume curves from well logs and saves the results under the selected well. It inverts the recorded log responses into the volumetric fractions of the rock-forming minerals plus pore space, using a constrained weighted least-squares solver.
The wizard can be run on a **single well** or in **batch mode** across many wells, and provides a live preview of the input logs, synthetic logs, stacked mineral volumes, misfit and quality flag before any curve is written to the project.

Purpose

The tool is designed for:

© 2026 Geomage 2003 LTD

- Estimating the volumetric fractions of rock-forming minerals from standard petrophysical logs
- Building a consistent lithology / mineralogy model for clastic, carbonate, evaporitic and organic-rich intervals
- Deriving total porosity together with the mineral volumes in a single closed inversion
- Generating synthetic logs, misfit and quality-control curves to validate the mineral model

Requirements

- At least one well with continuous well logs loaded in the **Data Manager**
- Recommended input curves: `RHOB` (bulk density), `NPHI` (neutron), `PEF` (photoelectric factor), `DT` (sonic slowness) and `GR` (gamma ray)
- An optional **VSH / VCL** curve when the clay/shale volume is to be fixed
- The more independent logs are supplied, the more minerals can be resolved reliably

Launching the Wizard

Open the wizard from the **Petrophysics** ribbon tab → **Execute** group → **Mineral Volume Estimation**.

The wizard opens as a dialog with a parameter panel on the left and an **Input/Output** preview panel on the right. The bottom of the dialog holds the **Save to workflow**, **Run**, **Save** and **Close** buttons.

Step-by-Step Guide

Selecting the Well and Interval

- Choose a well from the **Well** dropdown at the top of the panel.

© 2026 Geomage 2003 LTD

- To process several wells at once, enable **Batch calculation** and pick the scope from the adjacent list (for example **All wells** or a specific well group).
- Define the calculation interval on the depth-range tabs. The interval can be limited by **MD range**, **TVD range**, **TVDSS range**, or between **markers**. Depth units follow the project or well unit system.

Choosing the Calculation Method

Select a solver in the **Method** group. A short equation for the selected method is shown below the list.

- **Multi-mineral (constrained inversion)** – the default method. Estimates any number of enabled minerals plus porosity using constrained weighted least squares. Recommended for complex carbonates, mixed clastic-carbonate, evaporitic and shaly/organic-rich rocks.
- **Simple two-mineral** – estimates the ratio of exactly two minerals for quick QC (e.g. quartz/clay or calcite/dolomite). A dedicated **Porosity (Method 1)** group appears for this method, letting you set porosity to **None (relative fractions)**, **Fixed value**, or **From curve**.
- **Three-mineral linear** – solves exactly three components from the mapped logs using linear response equations.

Note: **Simple two-mineral** and **Three-mineral linear** require exactly two or three enabled minerals respectively; the wizard warns if the count does not match.

Mapping Input Logs and Weights

In the **Input Logs && Weights** table, map a well curve to each of the five log slots and set its relative weight in the inversion. The table has **Slot**, **Curve**, **Log type** and **Weight** columns.

- The five slots are `RHOB`, `NPHI`, `PEF`, `DT` and `GR`. A slot left blank is simply not used.
- Default weights are `RHOB` 1.0, `NPHI` 1.0, `PEF` 1.0, `DT` 0.7 and `GR` 0.5; edit the **Weight** cell to give a log more or less influence.

Mineral Set and Parameters

The **Minerals && Parameters** table defines which minerals take part in the inversion and their log responses. Columns are **Enabled**, **Name**, **ρ** (density), **Δt** (sonic slowness), **NPHI**, **PEF**, **GR**, **Vmin**, **Vmax** and **Colour**. The **ρ** and **Δt** column headers show the active units (g/cc or kg/m³, µs/ft or µs/m) according to the project unit system.

© 2026 Geomage 2003 LTD

The default mineral set is Quartz, Calcite, Dolomite, Clay, Anhydrite, Halite, Pyrite, Coal and Kerogen. You can:

- Enable or disable a mineral with its **Enabled** checkbox (only enabled minerals are solved and saved).
- Edit any parameter cell, set the **Vmin** / **Vmax** volume limits, and double-click the **Colour** cell to change the display colour.
- Add a mineral with **+ Add**, or reset edited rows with **Reset selected**.
- Load a predefined template from the **Load template** list – **Clean Sandstone**, **Shaly Sandstone**, **Carbonate**, **Evaporitic Carbonate**, **Organic-Rich Shale** or **Mixed Clastic-Carbonate**. A template populates the mineral list, parameters and recommended logs; all values remain editable.
- Save the current mineral set with **Save library…** and reload it later with **Load library…**.

Constraints

The **Constraints** group controls how the inversion is regularised:

- **Strict volume balance** – forces the sum of mineral volumes plus porosity to equal 1 (enabled by default).
- **Clip negative volumes to 0** – removes small negative volumes from the solution (enabled by default).
- **Use VSH / VCL curve** – fixes the clay/shale volume from a selected curve, solving the remaining minerals inside the clean-rock volume (disabled by default; enable it and pick the curve in the adjacent list).
- **φ max** – maximum allowed porosity (default 0.40).
- **Misfit threshold** – misfit above which a sample is flagged as poor quality (default 0.05).
- **Skip bad-hole intervals** – reserved for a future bad-hole flag curve; the option is present but not available in this version.

Output Settings

In the **Output** group:

- **Output suffix** – the suffix appended to every result curve name (default `_MM`).
- **Model name** – the name of the mineral model; the output curves are grouped under a LAS set named after it (default `Carbonate_MM_01`).
- **Curve overwrite** – how existing curves are handled: **Overwrite**, **Version** (default) or **Prompt**.

© 2026 Geomage 2003 LTD

Preview and Save

- Click **Run** to compute the model without writing any curves. The **Input/Output** panel then displays the input logs, synthetic logs, a stacked mineral-volume track, the misfit curve and the quality-flag track.
- Review the preview, adjust minerals, weights or constraints as needed, and re-run until the fit is satisfactory.
- Click **Save** to write the result curves to the well. In batch mode the model is applied to every target well and a summary report is shown afterwards.
- Use **Save to workflow** to store the configured job as a reusable workflow step, or **Close** to dismiss the wizard.

Results

When you click **Save**, the result curves are stored under the selected well, grouped in a LAS set named `Mineral volumes - <Model name>` (for example `Mineral volumes - Carbonate_MM_01`).
The following curves are created (names use the **Output suffix**, `_MM` by default). Only enabled minerals produce a volume curve:

- One mineral-volume curve per enabled mineral, e.g. `V_QUARTZ_MM`, `V_CALCITE_MM`, `V_DOLOMITE_MM`, `V_CLAY_MM` (v/v).
- `PHIT_MM` – total porosity (omitted when Method 1 uses relative fractions only).
- Synthetic logs `RHOB_SYNTH_MM`, `NPHI_SYNTH_MM`, `DT_SYNTH_MM`, `PEF_SYNTH_MM`, `GR_SYNTH_MM` reconstructed from the solved volumes.
- `MISFIT_MM`, `VOLUME_SUM_MM`, `QUALITY_FLAG_MM` and `DOMINANT_MIN_MM` quality-control curves.

Each output curve stores calculation metadata (method, solver, input logs and units, weights, minerals and parameters, constraints, interval, model name, date and software version). The `QUALITY_FLAG_MM` curve carries integer codes (0 = valid, and higher values for missing input, corrected volumes, high misfit and unstable solutions).
In batch mode a summary reports how many wells completed, how many were skipped for missing logs, and which finished with quality warnings.

Recommendations

© 2026 Geomage 2003 LTD

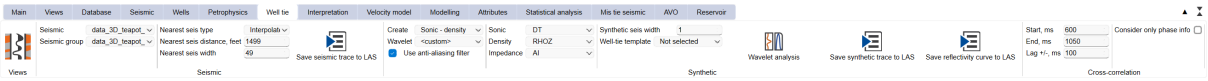
- Start from the closest **Load template** to your reservoir, then refine mineral parameters and enabled minerals.
- Keep the number of enabled minerals within what the available logs can resolve; too many minerals for too few logs makes the inversion unstable.
- Always **Run** and inspect the misfit and synthetic-log tracks before saving; high misfit usually means a wrong mineral set or mis-mapped logs.
- Avoid enabling two minerals with near-identical log responses; the wizard warns when responses are almost identical.
- Enable **Use VSH / VCL curve** when a reliable shale-volume curve already exists, to stabilise the clay estimate.
- Adjust the **Weight** of noisy logs downward rather than removing them entirely, so they still contribute to the fit.

See Also

- Petrophysics 49
- Porosity 65
- Saturation 69
- Shale Volume 58
- Lithology 77
- Workflows 535

3.1.7 Well tie bar

Well tie bar provides interface to perform seismic to well tie



For more information about well tie functionality please refer to Well tie view 287

Consists of 4 sections: Views, Seismic, Synthetic, Cross-correlation

The **Views** section opens a predefined set of views arranged for the well tie workflow: the **Well tie view** and the **Well tie cross-correlation view**. See Well tie view 287.

© 2026 Geomage 2003 LTD

The two views are added to the workspace that is active when you click the button. To build another set of well tie views, switch to another workspace or create a new one first and use the button there; you can keep several such sets in different workspaces.

If the well tie workspace itself is the active one, the button only brings it to the front and creates nothing, and if no well tie workspace exists yet, the first click creates it. Clicking the button again on a workspace that already holds these two views adds nothing - the views are not duplicated. A workspace that is still empty when you use the button takes the name **Well tie window**, while a workspace that already holds other views - a **Location map**, for example - keeps its own name.

In **Seismic** section it is possible to set seismic data for well tie.

Seismic	Choose the line which is near by to the well
Seismic group	Switch between different versions of loaded data
Nearest seis distance	Set searching distance of seismic data around the well
Nearest seis width	Set a number of traces in well tie view 287

The **Seismic** and **Seismic group** selections are stored for the selected well. Changing either drop down box saves that choice for the well selected at that moment, and when you select another well and come back, g-Space restores the seismic that well last used. The stored selections are saved with the project.

If a well has no stored selection yet, for example in a project saved before per-well selections were introduced, the selection currently shown is kept rather than reset. If the seismic stored for a well is no longer in the list for the current **Nearest seis distance**, the first entry in the list is selected.

In **Synthetic seismogram** section it is possible to set synthetic parameters:

Create	Set inputs: Sonic, Sonic - density, Impedance
Sonic, Density, Impedance	Drop down boxes allow you to choose the well logs
Synthetic seis width	Set a width in well tie view 287

© 2026 Geomage 2003 LTD

Well-tie template	Stores all well-tie templates created in g-Space
Wavelet analysis	Settings for the wavelet used for well-tie
Save reflectivity curve to LAS	Save the reflectivity series of the selected well as a well log

Save reflectivity curve to LAS writes the reflectivity series of the well currently selected in the Well tie view 287, computed from the same inputs as the synthetic seismogram. It is saved as a per-well LAS curve named **Reflectivity curve** inside a LAS entry named **ReflectivityLasCurve**, which appears under that well in the Data Manager and in the well log views. The curve is indexed by measured depth (**DEPT**, in metres) on a regular depth step, so you can review or export it like any other well log.

The well must have a checkshot and must already be set up for well tie, that is its well tie parameters must be stored, and the seismic interpretation licence is required. If a prerequisite is missing, g-Space shows an explanatory message - for example that the well has no checkshot - and saves nothing. Running the action again for the same well overwrites the curve in the same LAS entry instead of creating a second one, so versions do not accumulate.

In **Cross-correlation** section it is possible to set cross-correlation window parameters:

Start	Set start of the analysis interval
End	Set end of the analysis interval
Lag	Set a width of plot in the cross-correlation view

The **Start** and **End** values are stored for the selected well. When you select another well and come back, g-Space restores the cross-correlation window you last set for that well, in the same way as the **Seismic** and **Seismic group** selections of the **Seismic** section and the **Create**, **Sonic**, **Density** and **Impedance** selections of the **Synthetic seismogram** section. The stored windows are saved with the project.

If a well has no stored window yet, or the project was created before per-well windows were introduced, the values currently shown are kept rather than reset. Of the **Cross-correlation** settings, only **Start** and **End** are stored per well: **Lag** and **Consider only phase info** are single settings shared by all wells.

© 2026 Geomage 2003 LTD

Save calibrated T-D model

After you edit the checkshot anchors in the Well tie view 287, the **Save calibrated T-D model** action in the view toolbar saves the calibrated time-depth model for the selected well, built from the edited anchor positions.

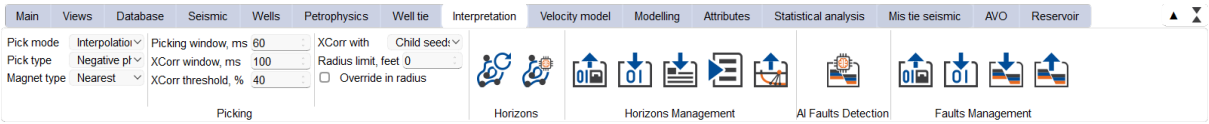
Two LAS curves are written under the corresponding well in the Data Manager and shown in the Well tie view 287:

- **TWT_calibrated** - the calibrated two-way time curve (ms)
- **Vint_calibrated** - the calibrated interval velocity curve (m/s)

Save calibrated T-D model action (outlined) in the toolbar of the Well tie view 287. The action is shown for a single well and is not available in the Well correlation view 296.

3.1.8 Interpretation

The **Interpretation bar** provides tools for seismic horizon and fault interpretation, including automatic picking, correlation control, AI-assisted detection, and import/export of interpretation objects.



The bar consists of the following groups: Picking, Horizons, Horizons Management, AI Faults Detection and Faults Management. For more information about the supported formats refer to Horizon Data Formats 513 and Fault Data Formats 515.

This bar brings together the horizon and fault interpretation tools that were previously divided between separate bars, so that picking, horizon map updates and the import and export of both horizons and faults are all available in one place. External topography is no longer loaded from this bar — it is now loaded from the **Topography** item in the Data Manager context menu.

Picking

Defines how horizons are picked in the seismic views. The available parameters are:

© 2026 Geomage 2003 LTD

- **Pick mode** — the picking mode (for example Autopicking).
- **Pick type** — the trace feature picked, such as Zero phase, peak, or trough.
- **Magnet type** — how the pick snaps to the seismic signal (for example Cross-correlation).
- **Picking window** and **XCorr window** (ms) — the time windows used for picking and for cross-correlation.
- **XCorr threshold** (%), **XCorr with**, **Radius limit** and **Override in radius** — control the correlation quality, the reference seed, and the radius within which picks are accepted.

These picking parameters are set directly in the bar and are also available in Module Properties 197.

A full description of the picking workflow is given in Seismic picking 586.

Horizons

Update the selected horizon maps from the current horizon picks.

Run the **AI Horizons Stack** wizard to automatically pick several horizons across the seismic volume using a trained model. To add the current settings to the active workflow without starting the calculation, select **Save to workflow**. When you open the saved job from the workflow, the wizard restores its settings; after editing them, select **Save** to update the job. See Workflows 535 for instructions.

Horizons Management

Import and export horizon interpretation data. For more information refer to Import horizons 438 / Export horizons 478.

Save horizon binary

Load horizon binary

Import horizon ASCII

Import horizon (charisma format)

© 2026 Geomage 2003 LTD

Export current horizon ASCII

AI Faults Detection

Runs AI-assisted fault detection on the selected seismic data to automatically highlight fault surfaces. See AI faults wizard 642 and Fault prediction 640 for the full workflow.

Faults Management

Import and export fault interpretation data. For more information refer to Import faults 441 / Export faults 480.

Export faults binary

Load faults binary — add faults from a binary file to the existing faults. If an imported fault name is already in use, g-Space appends _2, _3, and so on so that both faults are kept. If the file cannot be parsed, g-Space reports that it could not load the fault file and leaves the existing faults unchanged.

Import faults as charisma

Export faults as charisma

See Also

Seismic picking 586

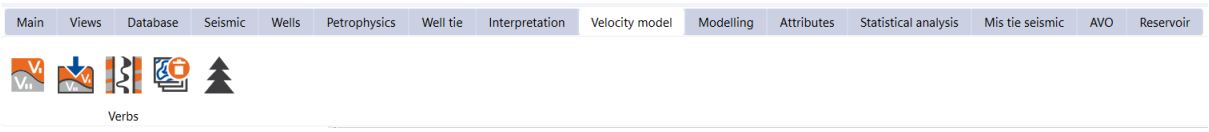
AI faults wizard 642

Ribbon bar 33

© 2026 Geomage 2003 LTD

3.1.9 Velocity model bar

The **Velocity model** bar gives you access to the tools used to build, import, calibrate and quality-control a depth velocity model. The velocity model is used to convert interpreted data between the time and depth domains and to support depth imaging and structural interpretation.



For more information about the velocity models please refer to Velocity model 658.

Buttons

The bar contains the following buttons:

Create depth velocity — opens the velocity modeling wizard, where you build a new depth velocity model (for example a constant or a layered model) from horizons, checkshots and well data.

Import depth velocity model — loads an existing depth velocity model from an external file into the project.

Update depth velocity by well tie — calibrates the depth velocity model to the wells, adjusting velocities so that the modelled depths match the well markers (well-tie).

Clear all well-tied maps — removes all well-tie corrections previously applied to the velocity maps, restoring the uncalibrated model.

Pine creation — builds a Pine chart from the velocity model for interval-velocity analysis and quality control.

© 2026 Geomage 2003 LTD

See Also

Velocity model 658

Import velocity model 678

Pine graph 332

Ribbon bar 33

3.1.10 Modelling

The **Modelling** bar provides tools for creating maps, building fault frameworks, and generating structural models from interpreted seismic and geological data.

The bar consists of the following sections: Interpolation, Map building, Map calculator, Fault Modeling and Structural Modeling.

Interpolation

Defines the interpolation settings used for map generation. The available parameters are:

- **Interpolation type** — the algorithm used to build the map. The available methods are **ABOS**, **Kriging** and **Adaptive Snap**.
- **Step X / Step Y** — the grid cell size of the resulting map.
- **Use internal points / Use polygons** — control whether internal control points and bounding polygons are taken into account during interpolation.
- **Fault clear radius, Taper size, Smooth aperture** — fine-tune how the map is shaped around faults and how strongly it is smoothed.

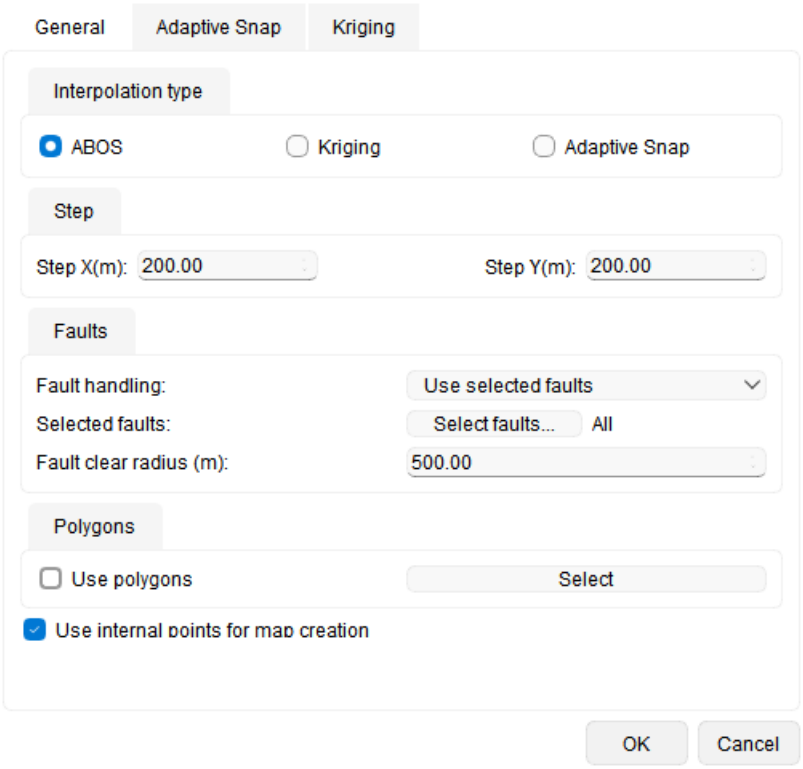
Set the interpolation type, grid steps, internal-point and polygon options, fault clear radius, taper size, and smooth aperture directly in the bar. Click **Settings** in the **Map calculator** group to open the interpolation settings, where you can choose **Fault handling** and configure method-specific options. The full set is also available from the **Module Properties** panel.

© 2026 Geomage 2003 LTD

Fault handling

The **Faults** group of the interpolation settings defines how faults are used when a map is built. Select one of the following in the **Fault handling** list:

- **Ignore faults** — faults are not taken into account and the map is interpolated continuously across them. The remaining fault controls are hidden.
- **Use selected faults** — click **Select faults...** and tick the faults to be used as barriers. The label beside the button shows how many faults are ticked, or **All** when the selection is empty and every fault of the project is used.
- **Use fault model** — choose one model in the **Fault model** list and use its surfaces as the barrier framework. Build the model in Fault Modeling 181 first; a model whose surfaces have not been built leaves the map without fault barriers.



With **Use selected faults** and **Use fault model** the map is interpolated separately on each side of a fault, so values are not smoothed across it, and **Fault clear radius** becomes available. Input data lying closer to a fault than this distance is discarded and that strip is rebuilt from the reliable data further away, which removes noisy picks along the fault without leaving the strip empty. With a radius of 0 the input data is used as it is, and only the grid cells crossed by a fault separate its two sides. Only faults and models of the same domain as the map, time or depth, are applied.

© 2026 Geomage 2003 LTD

Where a fault classified as **Normal** crosses the grid, the cells along its trace stay empty, so the map is interrupted along the fault line; faults of the other types keep the map continuous. See Faults 158 for assigning the fault type.

The mode, the selected faults or model, and the clear radius are stored with the interpolation settings, so every action that creates or recreates maps from horizons, points or markers uses them.

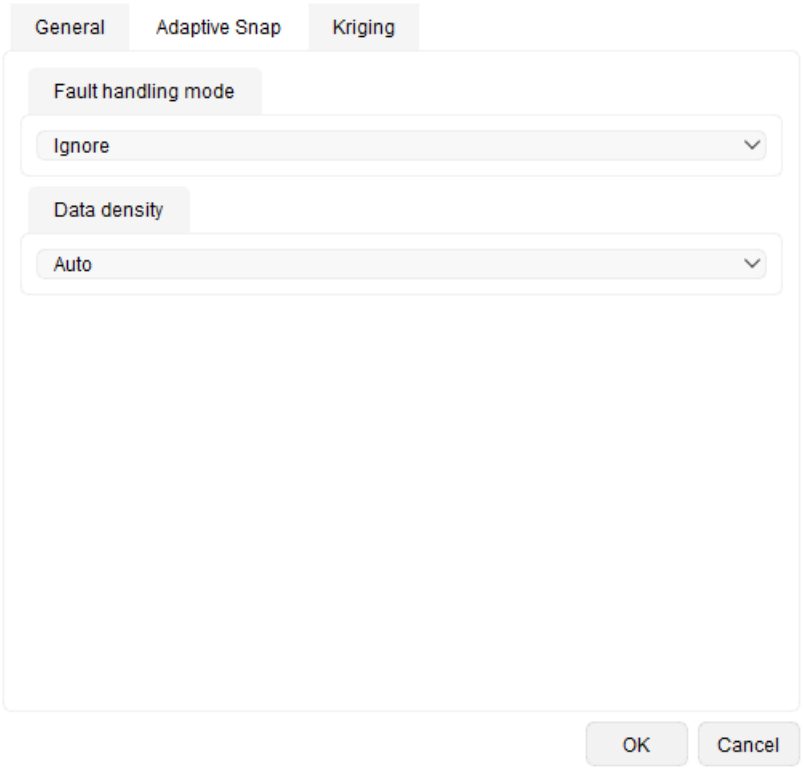
Adaptive Snap

Adaptive Snap is a convergent, coarse-to-fine interpolation method. It builds the map from the input data over successive refinement levels and snaps the final surface onto the input control points, so the resulting map honours the picked data. Mixed input data (points and interpreted horizons) can be used together as control data, and faults can be applied as constraints to preserve structural discontinuities.

When **Adaptive Snap** is selected, the following additional parameters are available on the **Adaptive Snap** tab of the interpolation settings dialog:

- **Fault handling mode** — controls how faults constrain the interpolation. The options are **Ignore** (the map is continuous across faults), **Barrier** (faults block the exchange of values between their two sides, preserving discontinuities) and **Final refinement only** (faults are ignored during the early levels to keep the regional trend and applied only in the final pass to introduce local discontinuities).
- **Data density** — tells the method how the input control data is distributed, so that it can adapt the way values are smoothed between the control points. **Auto**, the default, derives this from the spacing of the input data and the grid step. Select **Sparse (wells)** when the map is built from a few widely spaced points, such as well markers, or **Dense (grid)** when the input data is closely spaced, such as an interpreted horizon grid.

© 2026 Geomage 2003 LTD

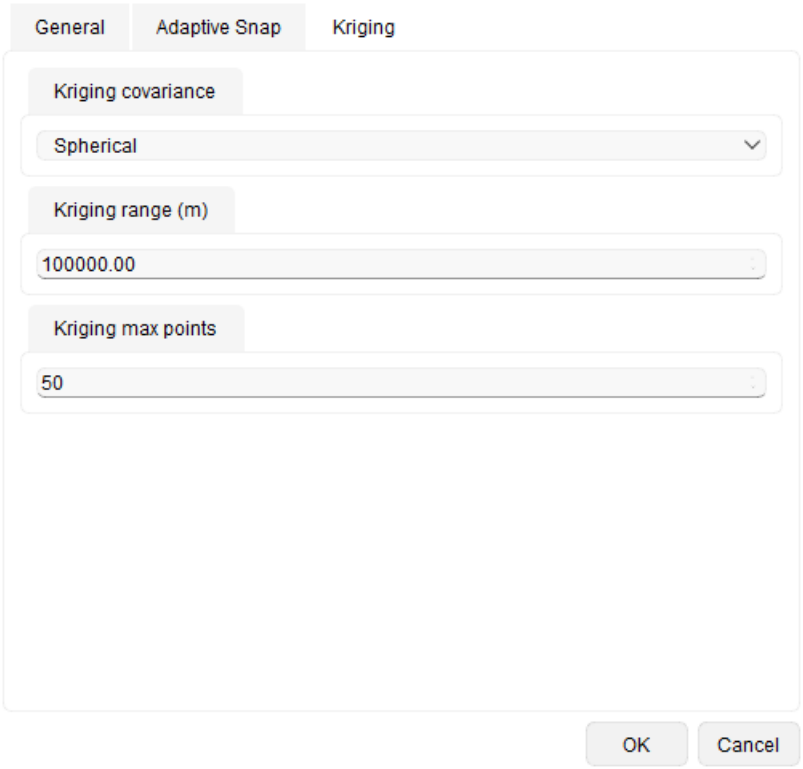


Kriging

When **Kriging** is selected, each node of the map is estimated from the nearest input points, weighted by a covariance model. The following additional parameters are available on the **Kriging** tab of the interpolation settings dialog:

- **Kriging covariance** — the covariance model that defines how the influence of an input point falls off with distance. The options are **Spherical**, **Gaussian** and **Exponential**.
- **Kriging range** — the correlation distance of that model. Input points lying far beyond this distance from a node contribute practically nothing to its value. The group is labelled **Kriging range (m)** or **Kriging range (ft)**, following the measurement system of the project.
- **Kriging max points** — how many of the nearest input points are used to estimate each node.

© 2026 Geomage 2003 LTD



For more information about map building please refer to Map creating 724.

Map building

Holds the single **Map building** command, which opens the **Map building** dialog. The dialog builds a map from a horizon, from well markers or from a point set, using its own set of parameters, and it stays open while you work, so maps can be built one after another.

Map calculator

Performs arithmetic and logical operations on maps (depth maps, velocity maps, and others) to create new calculated map outputs based on user-defined expressions.

The group contains **Settings**, **Open map calculator**, **Calculate isopach**, and **Residual calculation**.

© 2026 Geomage 2003 LTD

Open map interpolation settings

Open map calculator

Fault Modeling

Builds a fault framework from interpreted fault data to support structural analysis and geological model construction.

Structural Modeling

Creates a structural model using horizons, faults, and maps to represent the main geological framework of the reservoir. See Structural modeling 798 and Geological modeling 758 for the full workflow.

See Also

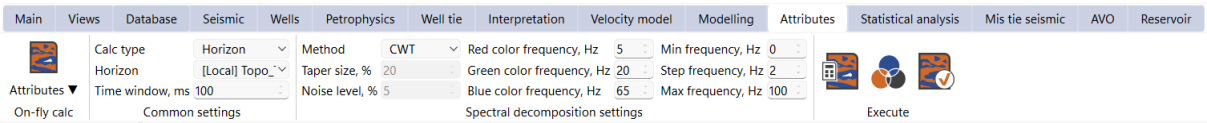
Map creating 724

Structural modeling 798

Ribbon bar 33

3.1.11 Attributes bar

Attributes bar allows you to generate/calculate the seismic attributes for further analysis during the seismic interpretation.



© 2026 Geomage 2003 LTD

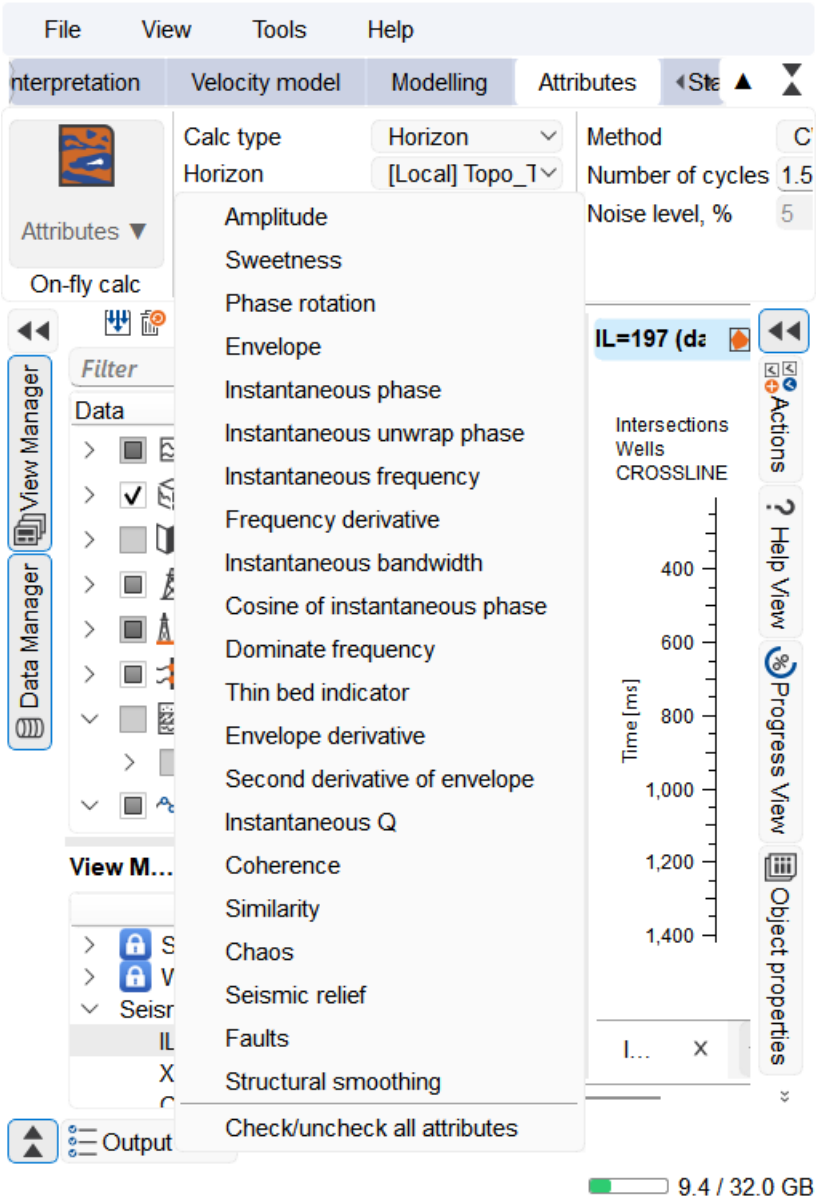
For more information about the attributes calculation please refer to chapters Seismic attributes 680 and Spectral decomposition 688

Consists of 4 sections: On-fly calc, Common settings, Spectral decomposition settings, Execute.

On-fly calc section:

Consists of one button which opens a context menu. In this menu you can select attributes for maps calculation or select an attribute to see it on the seismic section:

© 2026 Geomage 2003 LTD

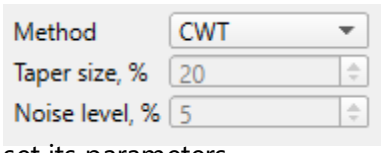


In **Common settings** section you can choose parameters of calculation**:**

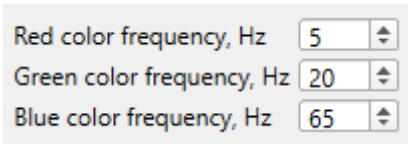
Calc type	Choose type of calculation between horizon or whole seismic dataset
Horizon	Choose horizon for calculation
Time window, ms	Set time window of calculation

© 2026 Geomage 2003 LTD

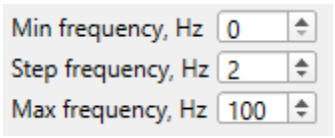
In **Spectral decomposition settings** section you can choose parameters for this attribute**:**



Set method of spectral decomposition calculations and set its parameters



Set colors for each frequency



Set boundary parameters for calculations

Execute section:

Open the window for attributes calculation settings and start of calculation

Run spectral decomposition

Run QC for selected horizon picking. This icon is associated with spectral composition to QC and select the parameters from spectral decomposition from the selected horizon

The following list provides a detailed explanation of the seismic attributes available in g-Space. For each attribute, a brief theoretical background is given along with its practical application in seismic data analysis.

© 2026 Geomage 2003 LTD

Amplitude: Represents the strength or intensity of the seismic signal at each reflection point. Amplitude variations are often related to changes in rock properties, fluid content, or lithology, making it a critical parameter for identifying potential reservoirs, stratigraphic boundaries, and geological features.

Phase rotation: Rotates the phase of the seismic trace by a constant angle and outputs the rotated trace. It is typically used to correct a residual phase shift in the data, or to check how a different phase affects the appearance of the reflections. Set the angle with the **Rotation angle** parameter, in degrees from -180 to 180, where 0 leaves the trace unchanged. It is available in the **Phase rotation attribute parameters** group of the **Advanced** section in Module Properties 197.

Envelope: The envelope is a smooth curve that outlines the peaks and troughs of a seismic wave. It's often used to detect changes in seismic reflectivity, which can be linked to stratigraphic features, faults, or lithological variations. The envelope highlights zones of high energy and can improve the interpretation of complex seismic events.

Instantaneous Phase: The instantaneous phase attribute measures the phase angle of the seismic signal at any given point in time. This attribute is phase-independent, meaning it is not influenced by amplitude variations, making it useful for identifying continuous horizons and tracking lateral continuity in the subsurface.

Instantaneous Unwrap Phase: This is an unwrapped version of the instantaneous phase, where discontinuities in phase values are removed, providing a continuous phase profile. It helps in mapping stratigraphic layers more precisely, especially in areas where reflectors might be obscured by phase wrapping.

Instantaneous Frequency: This attribute calculates the frequency of the seismic signal at each point in time. It highlights changes in frequency that may correspond to subsurface geological variations such as layer thickness, rock types, and depositional environments. Instantaneous frequency is often used for high-resolution stratigraphic interpretation.

Frequency Derivative: The frequency derivative shows the rate of change of frequency over time. Rapid changes in frequency can indicate transitions between different rock types or depositional environments. It helps in identifying seismic features such as unconformities, thin beds, or channel boundaries.

Instantaneous Bandwidth: Instantaneous bandwidth represents the range of frequencies present at any point in the seismic signal. A higher bandwidth often indicates more complex geological structures, while a lower bandwidth can be associated with simpler, homogeneous layers. This attribute is useful for characterizing reservoir quality and seismic resolution.

© 2026 Geomage 2003 LTD

Cosine of Instantaneous Phase: The cosine of the instantaneous phase converts phase information into a continuous waveform, emphasizing differences in phase between neighboring traces. This attribute is valuable for identifying faults, stratigraphic discontinuities, and other structural features that may be missed with traditional amplitude-based interpretation.

Dominant Frequency: Dominant frequency refers to the most prevalent frequency in the seismic signal. It can indicate the average thickness of the subsurface layers and is often related to depositional environments. A lower dominant frequency suggests thicker layers, while a higher frequency points to thinner beds or more rapid changes in geology.

Thin Bed Indicator: This attribute is designed to identify the presence of thin beds that are smaller than the resolution of the seismic wavelength. By analyzing amplitude and frequency variations, it highlights subtle features that might not be clearly visible in standard seismic attributes, helping to detect important stratigraphic details in thinly bedded reservoirs.

Envelope Derivative: The envelope derivative measures the rate of change in the envelope of the seismic signal. It is useful for pinpointing areas where there is a significant transition in reflectivity, which may correspond to abrupt lithological changes, fault zones, or sharp depositional boundaries.

Second Derivative of Envelope: The second derivative of the envelope further enhances areas of rapid change in reflectivity. This attribute is particularly useful for detecting fine geological features that may be missed with other attributes, such as thin layers or subtle faults.

Instantaneous Q: Instantaneous Q represents the quality factor of the seismic wave, which measures the attenuation of the signal as it travels through the subsurface. High Q values suggest lower attenuation (indicating harder or more solid rock), while low Q values indicate higher attenuation (associated with softer rock or the presence of fluids). This attribute is often used to infer rock porosity or fluid content in reservoirs.

Coherence: Coherence measures the similarity of seismic traces across a specified window, helping to highlight discontinuities in the subsurface such as faults, fractures, or stratigraphic boundaries. It is a powerful tool for fault interpretation and for identifying subtle geological structures that may not be visible in standard amplitude data.

Similarity: Similarity compares seismic traces within a neighborhood and highlights differences in waveform shapes. This attribute is often used to identify features such as faults, channels, and stratigraphic boundaries, as well as variations in rock type or fluid content.

Chaos: The chaos attribute detects areas where the seismic signal appears disordered or non-coherent, often corresponding to zones with complex geological structures such as fault

© 2026 Geomage 2003 LTD

zones, fractured reservoirs, or highly heterogeneous formations. It can help interpret regions where seismic reflections are disrupted or unclear.

Seismic Relief: Seismic relief highlights the vertical variation in subsurface reflector depth, providing a clear representation of topography and structural variations. It is useful for identifying features such as anticlines, synclines, fault blocks, and other geological structures that involve significant elevation changes in the subsurface.

3.1.12 Statistical analysis bar

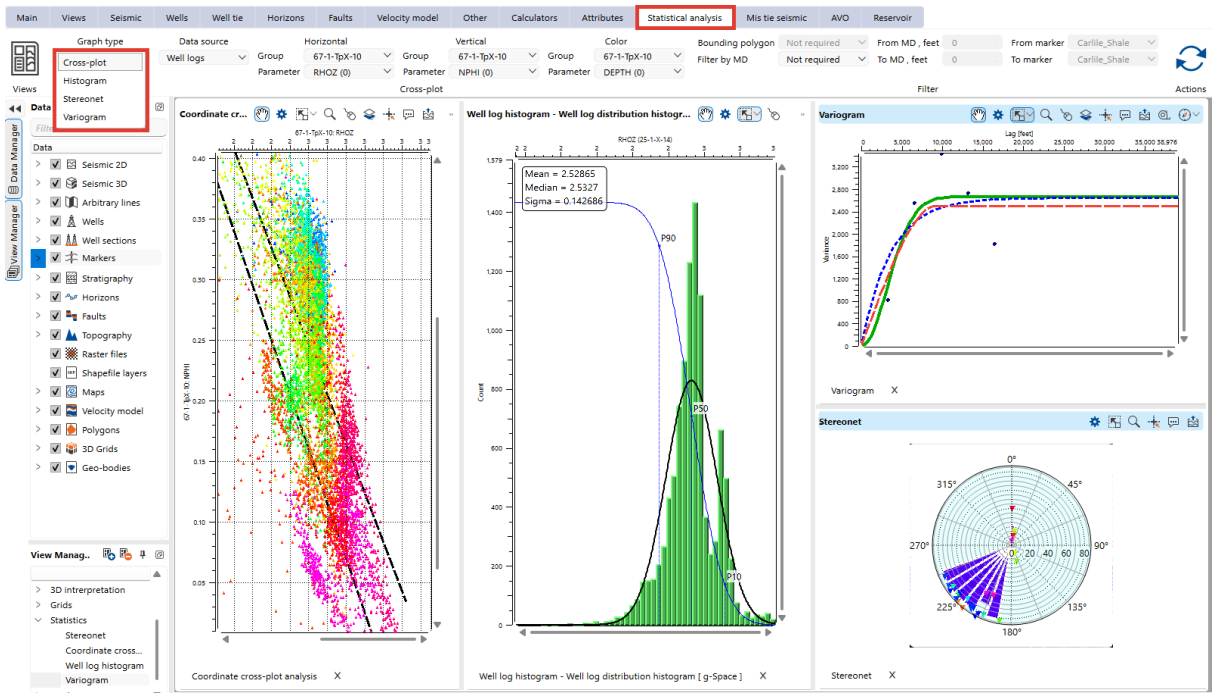
Statistical analysis view in g-Space is a powerful tool for analyzing and visualizing relationships between different attributes.

It provides the following graph types:

1. **Cross-plot**
2. **Histogram**
3. **Variogram**
4. **Stereonet View**
5. **Pine graph**
6. **Tornado**

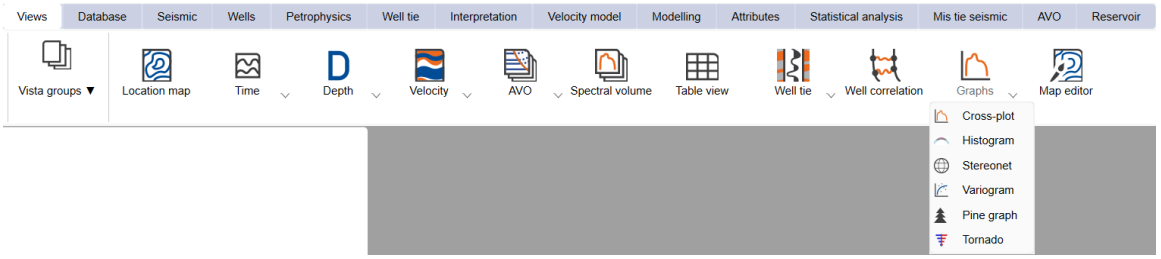
Each graph type serves a unique purpose in analyzing subsurface data and identifying trends, patterns, or distributions.

© 2026 Geomage 2003 LTD



To access these Views:

1. Navigate to the Views Bar 38 in the Ribbon.
2. Select the **Graphs** option.
3. Open the desired graph type: Cross-plot 330, Histogram 327, Variogram 335, Stereonet 336, Pine graph 332 or Tornado View.



In the **Statistical Analysis Bar** users can configure their plots.

Cross-Plot view is used to visualize the relationship between two selected attributes (X and Y axes). This is particularly useful for identifying correlations or patterns between the two variables.

Data Source: Choose the type of data to analyze (e.g., Well Logs or XY Data).

Horizontal Axis: Select the **Group** and corresponding **Parameter** for the X-axis.

© 2026 Geomage 2003 LTD

Vertical Axis: Select the **Group** and corresponding **Parameter** for the Y-axis.

Value filter: For the Well logs data source, the Horizontal and Vertical parameters can each be limited to a From / To value range. The full procedure is described in Cross plot 745.

Color: Optionally assign a parameter for point coloring to enhance visualization.

Bounding Polygon: Limit points displayed on the cross-plot using a predefined polygon.

Filter by MD/Markers: For well data, filter points within specific measured depth (MD) ranges or between selected markers.

Histogram view provides a graphical representation of the distribution of an attribute, giving users insights into the frequency and range of data values.

Data Source: Choose the type of data to analyze (e.g., Well Logs or XY Data).

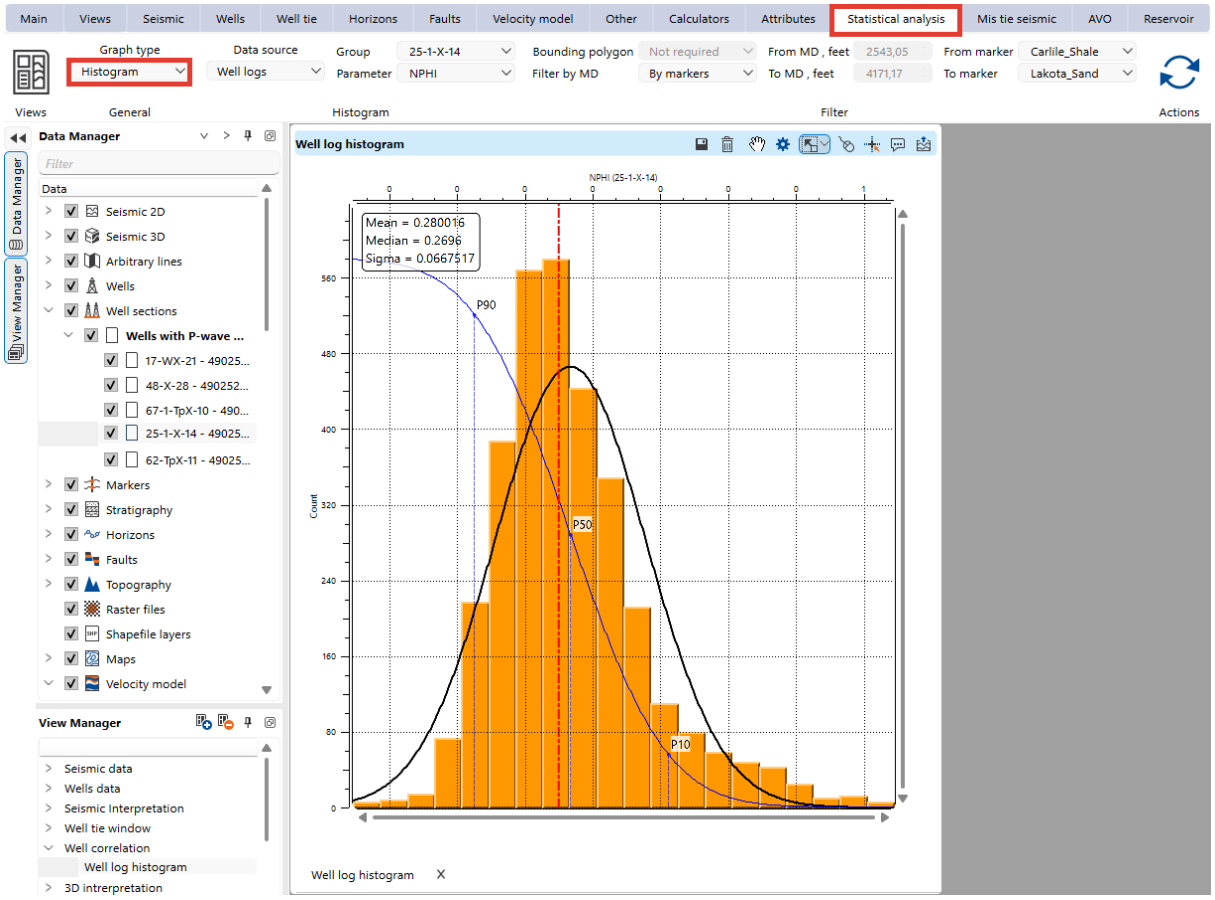
Group: Select the well log group to be analyzed.

Parameter: Choose the parameter to visualize.

Bounding Polygon: Limit points displayed on the cross-plot using a predefined polygon.

Filter by MD/Markers: For well data, filter points within specific measured depth (MD) ranges or between selected markers.

© 2026 Geomage 2003 LTD



The **Variogram View** in g-Space is a powerful tool for analyzing the spatial continuity and correlation of subsurface attributes. It helps users evaluate the variability of geological properties over different distances and directions.

Data Source: Choose the type of data to analyze (e.g., Well Logs, Well Attributes, Grid Maps, Horizons, Markers, Topography).

Data name: Choose the specific parameter to visualize spatial correlation.

Azimuth step, degree: Defines the angular interval for directional variogram calculations. A smaller step provides more detailed anisotropy analysis, while a larger step smooths directional variations.

Lag step: Determines the spatial separation between data points used in the variogram calculation. A smaller lag step provides finer resolution, while a larger lag step captures broader trends.

Lag max: Specifies the maximum distance to be analyzed. It defines how far the variogram extends before reaching the sill (where spatial correlation becomes negligible).

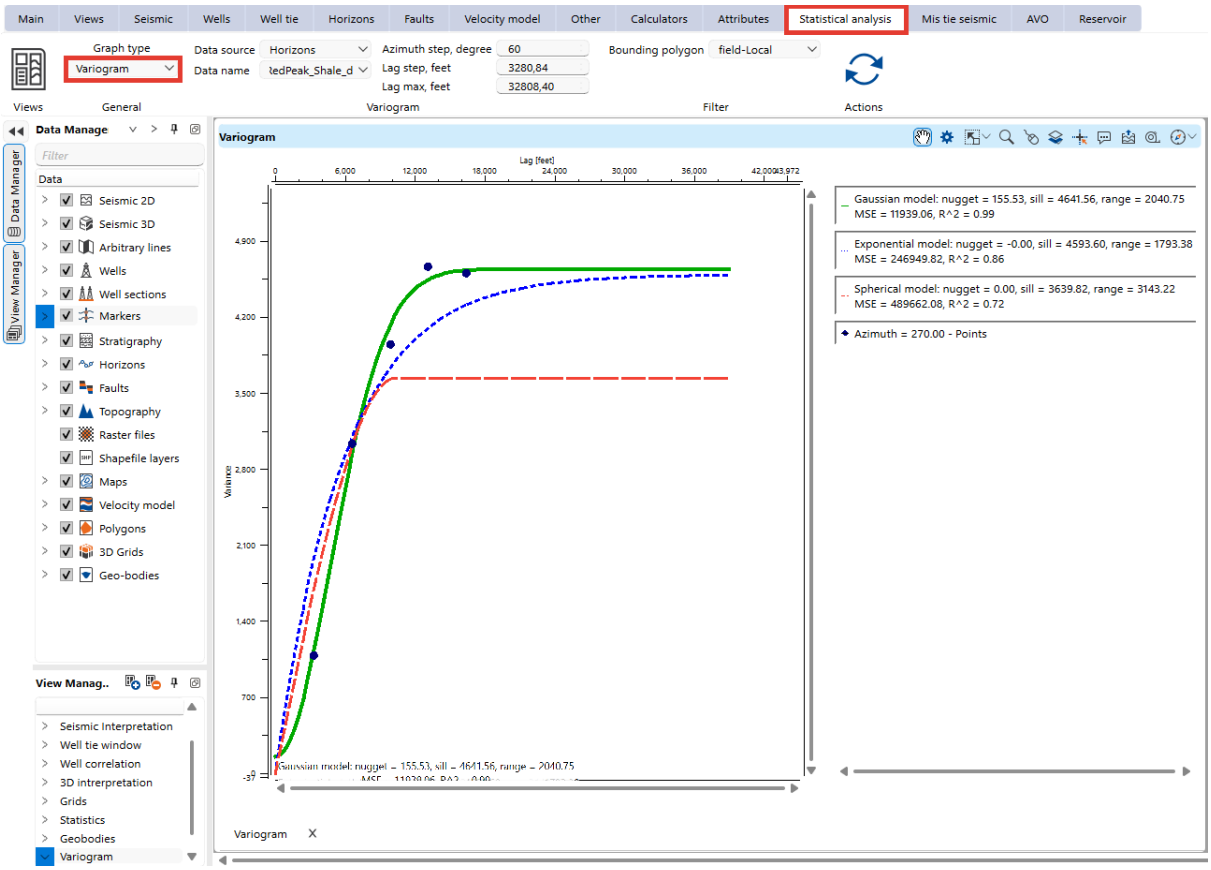
Value extraction: Determines how values are sampled for variogram computation. Here are the **Value Extraction** options for the **Variogram View** in g-Space:

© 2026 Geomage 2003 LTD

1. **Average** – Computes the mean value of the selected attribute within the defined lag step.
2. **RMS (Root Mean Square)** – Calculates the square root of the average of the squared values, emphasizing higher values in the dataset.
3. **Minimum** – Extracts the lowest value within the defined lag step.
4. **Maximum** – Extracts the highest value within the defined lag step.
5. **Dominant** – Selects the most frequently occurring value within the lag step.
6. **Median** – Determines the middle value of the dataset, reducing the influence of outliers.

Depth mode: For the Well logs data source, limits the depth range used for the variogram analysis: Between MD (the default) uses the Start MD and End MD depths, Between markers uses a Start marker and End marker, and Between horizons uses a Start horizon and End horizon. It appears only when the data source is Well logs.

Bounding Polygon: Limit points used in the variogram calculation by defining a spatial boundary.



© 2026 Geomage 2003 LTD

The **Stereonet** view is used to visualize the orientation of structural data, such as dip and azimuth, on a stereographic projection.

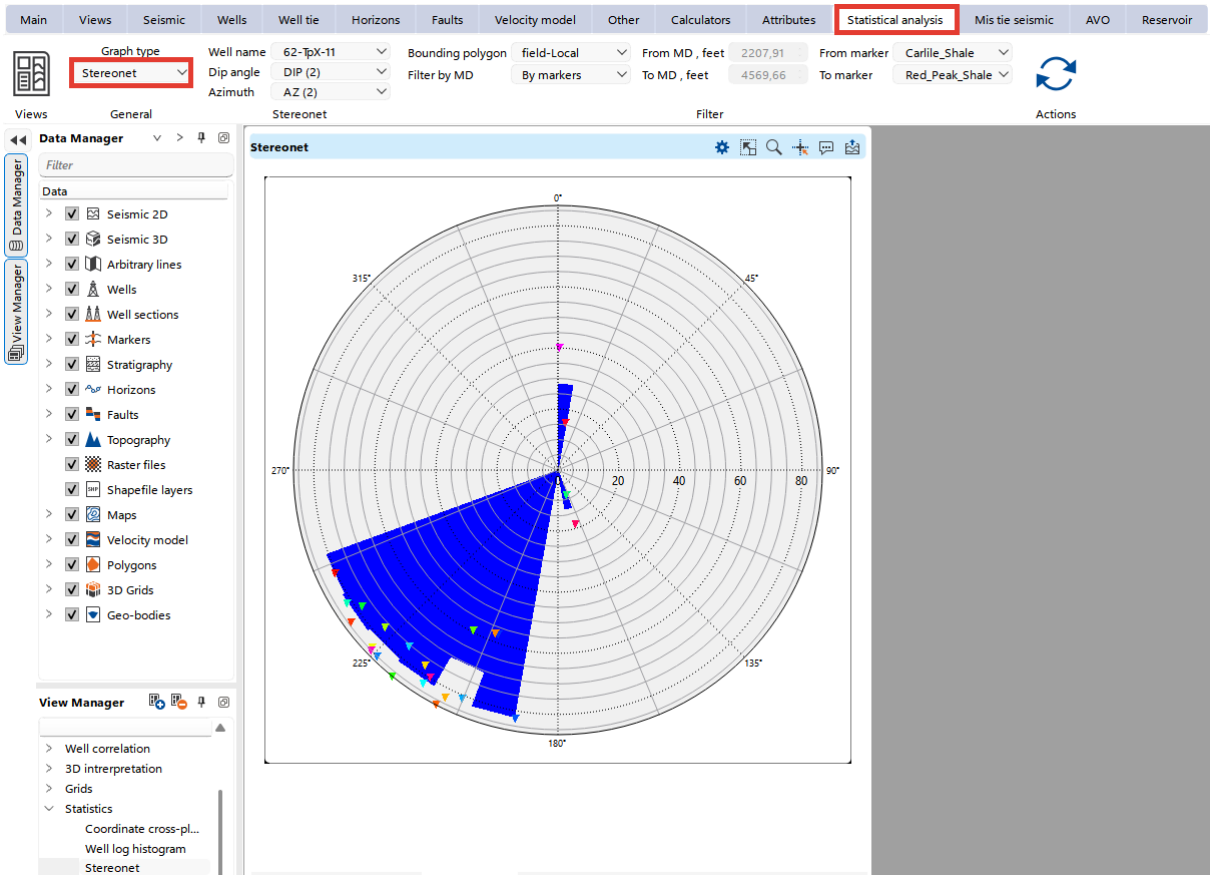
Well Name: Select the well to be analyzed.

Dip Angle: Choose the well log parameter for dip.

Azimuth: Select the well log parameter for azimuth.

Bounding Polygon: Limit points displayed on the cross-plot using a predefined polygon.

Filter by MD/Markers: For well data, filter points within specific measured depth (MD) ranges or between selected markers.



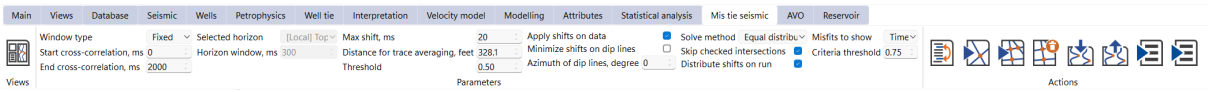
click **Update** to see updated results after changing settings.

For more information about the cross-plotting please refer to Statistical analysis 742

© 2026 Geomage 2003 LTD

3.1.13 Mis tie seismic bar

Mis tie seismic bar allows you to find out and reduce the time difference or phase difference between two intersecting lines.



For more information about miss ties please refer to Miss-tie correction 713

Consists of 3 sections: Views, Parameters, Actions

Views section:

Create mis-tie workspace for analysis line intersections

In the **Parameters** section it is possible to define the parameters of time and phase mis-ties calculation:

- **Window type:** choose fixed time window or time window around horizon
- **Start/End cross correlation:** choose start and end values for fixed time window
- **Selected horizon:** choose horizon using for center of time window. The horizon must already have a map. If the selected horizon has no map, mis tie analysis does not run and asks you to create a map for this horizon first, see Map creating 727
- **Horizon window:** choose time window around horizon
- **Max shift:** maximum shift that can be applied to seismic data
- **Distance for trace averaging:** choose distance for averaging traces
- **Threshold:** choose threshold value for traces to be averaged
- **Apply shift on data:** apply existed shifts to seismic data in seismic visualization. When the box is unchecked seismic data visualized without any corrections
- **Minimize shifts on dips lines:** this option allows to reduce the shifts of lines located along the structure slope. Such lines may be more reliable because of better migration results
- **Azimuth of dip lines:** set azimuth of lines in which shifts will be reduced
- **Solve method:** choose math method for mis-ties minimization

© 2026 Geomage 2003 LTD

- **Skip checked intersections:** option allows not to use checked intersections in automatic miss-ties calculation
- **Distribute shifts on run:** apply automatically calculated shifts to seismic data. When the box is unchecked mis-ties will be visualized on the location map only
- **Misfits to show:** choose phase of time type of miss-ties for visualization
- **Criteria threshold:** level of cross-correlation used in the automatic miss-ties calculation

In the **Actions** section it is possible to execute/generate the time and phase misties:

Set seismic group priority

Run for current intersection

Run for all intersections

Clear all mis-tie corrections

Import cross points table from ASCII

Export cross points table to ASCII

Auto fit volumes

Calculate volume intersection

Copying the mis-tie table

The cross points table calculated for the line intersections is shown as a table in the mis-tie workspace, and its contents can be copied to the clipboard. Click the **Copy** button in the table

© 2026 Geomage 2003 LTD

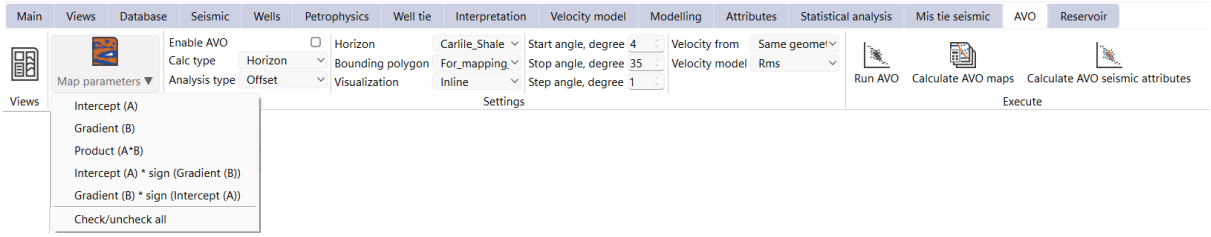
toolbar to copy the whole table, including the column headers, as tab-separated text, or select cells in the table and press **Ctrl+C** to copy only the selected cells. The copied data can be pasted directly into a spreadsheet. The table can also be written to a file with **Save as CSV**.

3.1.14 AVO bar

AVO (Amplitude Variation with Offset or Amplitude Versus Offset)

The AVO analysis allows users to measure rock properties and identify potential hydrocarbon reservoirs by analyzing how seismic reflection amplitudes change with the distance (or offset) between the source and receiver.

The user can measure the rock properties using AVO.



To start working with AVO, you need to create a dedicated workspace for AVO analysis. This workspace will allow you to manage and visualize the various aspects of AVO data.

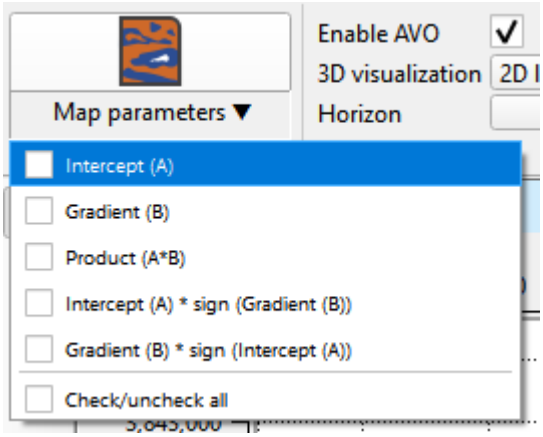
Once created, the AVO workspace will provide tools and views specifically designed for AVO analysis, such as cross-plot views and attribute maps.

Create and open AVO analysis workspace

Map parameters.

The **Map Parameters** section for AVO analysis, as shown in the interface, allows the user to select and calculate specific attributes like the **Intercept (A)**, **Gradient (B)**, and their products.

© 2026 Geomage 2003 LTD



Here's a description of the options in the "Map Parameters" menu:

1. **Intercept (A)** represents the amplitude at zero offset, essentially capturing the baseline reflection coefficient. It helps in identifying the type of lithology or fluid content in the subsurface.
2. **Gradient (B)** reflects the rate of change of amplitude with offset. The gradient provides additional details about the fluid content and type of rock present, particularly useful in detecting gas zones or differentiating between fluids.
3. **Product (A * B)** is a combination of the intercept and gradient, highlighting areas where both factors play a significant role together. It can be useful in enhancing certain anomalies within the seismic data.
4. **Intercept (A) * sign(Gradient (B))** applies a sign function to the gradient, combining it with the intercept to highlight changes in polarity and detect possible hydrocarbon indicators.
5. **Gradient (B) * sign(Intercept (A))**, similar to the previous option, but the sign function is applied to the intercept. This variation emphasizes areas where the intercept's polarity affects the gradient, which can be useful for identifying fluid contacts.
6. **Check/uncheck all** allows users to select or deselect all the parameters for processing at once.

Settings section:

- **Enable AVO:** This option must be checked to activate AVO processing within the workspace
- **Calc. type:** Choose calculation type between horizon or whole seismic. Perform AVO analysis along a selected horizon, focusing on specific geological layers. Apply AVO analysis across the entire seismic dataset, providing a broader view of the subsurface.

© 2026 Geomage 2003 LTD

- **Analysis type:** Choose whether to use **Offsets** (the distance between the seismic source and receivers) or **Angles** (incident angles of seismic waves) for your calculations. This selection impacts how the AVO attributes are derived.
- **Horizon:** Select the horizon that you wish to analyze. This is particularly important when you are focusing on specific stratigraphic layers or intervals.
- **Bounding polygon:** If you need to limit the AVO analysis to a particular area, you can define a bounding polygon.
- **Visualization:** By default None. The user can select 2D line in case of 2D dataset, inline/cross line for 3D dataset
- **Start/stop/step angle:** This setting allows you to define the range and step of incident angles used in pre-stack data visualization. Adjusting these angles can help in analyzing different AVO responses across various offset ranges.
- **Velocity from:** Select the velocity source used to convert offsets into incidence angles: **Same geometry** (velocity data stored with the same geometry as the analyzed seismic), **Volume** (a velocity volume), **Velocity model** (a velocity model of the project) or **Constant** (a fixed velocity of 2000 m/s). By default, **Constant** is selected.
- **Velocity model:** Select the velocity to be used with the chosen source. The available items depend on **Velocity from**: **Rms** or **Interval** for **Same geometry**, the velocity volumes of the project for **Volume**, the velocity models of the project for **Velocity model**, and the fixed value **2000 m/s** for **Constant**.

Execute section**:**

Run AVO cross plotting

Calculate AVO attributes maps

Calculate AVO attributes 3D volume or 2D sections

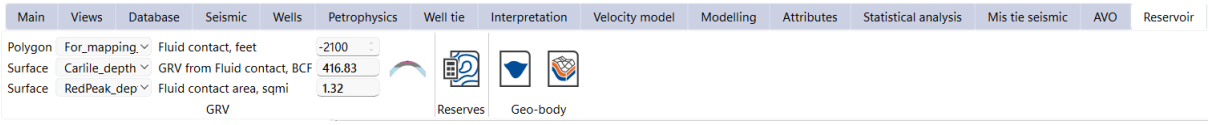
The AVO analysis in g-Space is a comprehensive toolset that enables detailed examination of seismic data to evaluate subsurface properties. By carefully configuring the settings and executing the appropriate analysis, users can gain valuable insights into reservoir characteristics and make informed decisions about exploration and development.

For further information about specific AVO functions and settings, please refer to AVO 700

© 2026 Geomage 2003 LTD

3.1.15 Reservoir bar

Reservoir bar allows you to perform volumetric calculations.



For more information about functions please refer to Volumetric calculations 790

Consists of 3 sections: GRV, Reserves, Geo-body

GRV section allows to calculate gross rock volume:

Choose a **Polygon**, select the top depth map in the first **Surface** list, and optionally select a bottom depth map in the second **Surface** list. Leave the second list set to **Not required** when the fluid contact is the lower boundary.

Set the **Fluid contact** depth. The gross rock volume is calculated from the selected depth maps and fluid contact on the maps' own grid, so no separate top or bottom levels, step, bar count or regridding settings are required.

Execute the gross rock volume calculation. The **GRV from Fluid contact** value is displayed next to the button.

Reserves section opens the **Calculate reserves** wizard for effective reservoir volume, oil, gas and CO2 estimates:

Open the **Calculate reserves** wizard

The wizard offers **Simple** and **Advanced** (stochastic) modes and can take the reservoir volume from surface maps or from a project grid. See Volumetric calculations 790 for details.

© 2026 Geomage 2003 LTD

Geo-body section creates a geo-body from the current GRV boundaries or opens the 3D geological model workflow:

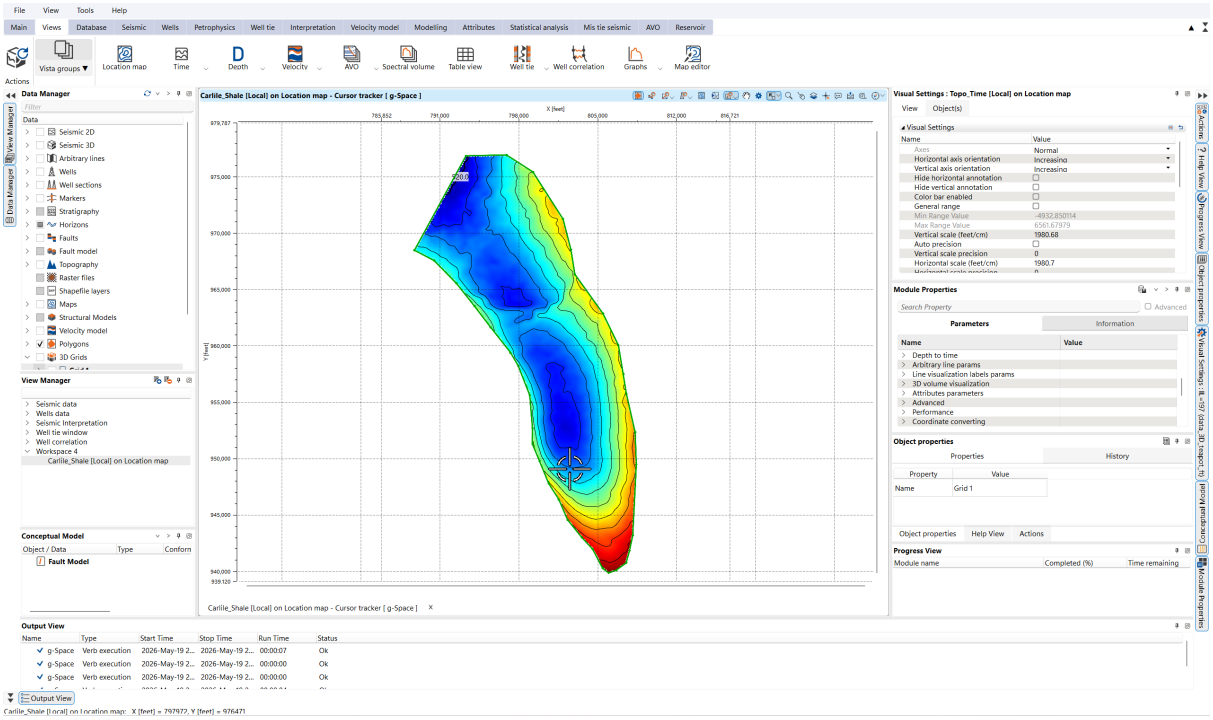
Create geo-body from gross rock volume

The **Create geo-body from gross rock volume** action uses the **Polygon**, the selected **Surface** depth map or maps, and the **Fluid contact** depth from the **GRV** section.

Create 3D geological model

3.2 Panels

In this chapter you will get the general introduction to panels in g-Space



General information:

Panels are located in the left and right borders and on the bottom of the g-Space window and are combined into panels sets. By default, the Data Manager 124, View Manager 189 and

© 2026 Geomage 2003 LTD

Conceptual Model 191 are placed on the left and are used mostly for viewing and storing your data. On the right panels set you will find the Module Properties 197, Visual Settings 206, Progress View 221, Actions 214 and Object properties 221 which are used mostly for project settings and data manipulation. Output View 219 is located on the bottom.

Setting up the panels:

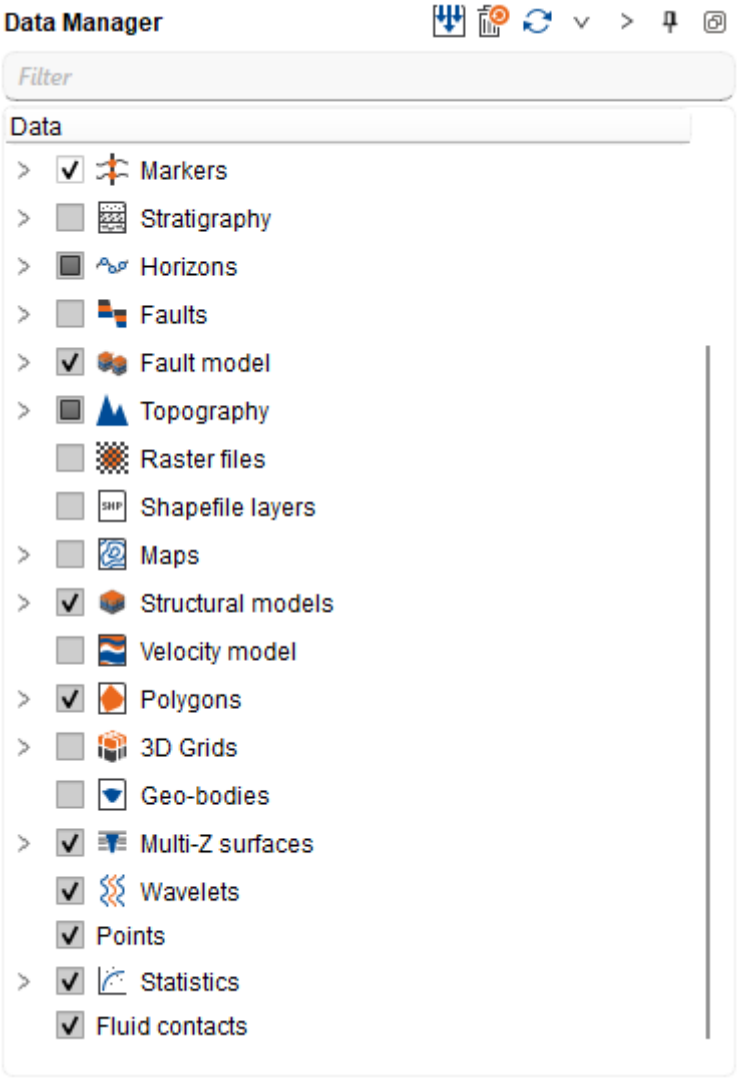
You can organize the panels according to your work habits. To do this drag them by holding left button mouse on the name of the panel.

To hide or extend panels you need to click on double arrows on the panels sets: , or

3.2.1 Data Manager

Data manager provides access to any data object from the project database and allows to control its creation, update, delete, visibility and operations

© 2026 Geomage 2003 LTD



Icons and visibility

The checkbox and icon next to each data object show whether it is – or can be – displayed in the active view:

- **Checkbox checked** – the data is displayed in the active view window.
- **Transparent icon** – the data can be visualized in the active window.
- **Gray icon** – the data cannot be visualized in the active window type.
- **Expand arrow** (>) – click to expand or collapse nested data categories.

Two buttons at the top of the panel change visibility for many objects at once. **Show all** and **Hide all** are placed to the left of **Sync visibility**: **Show all** ticks every object that can be displayed in the active view, and **Hide all** clears the checkbox of every object currently shown there, hiding them all at once.

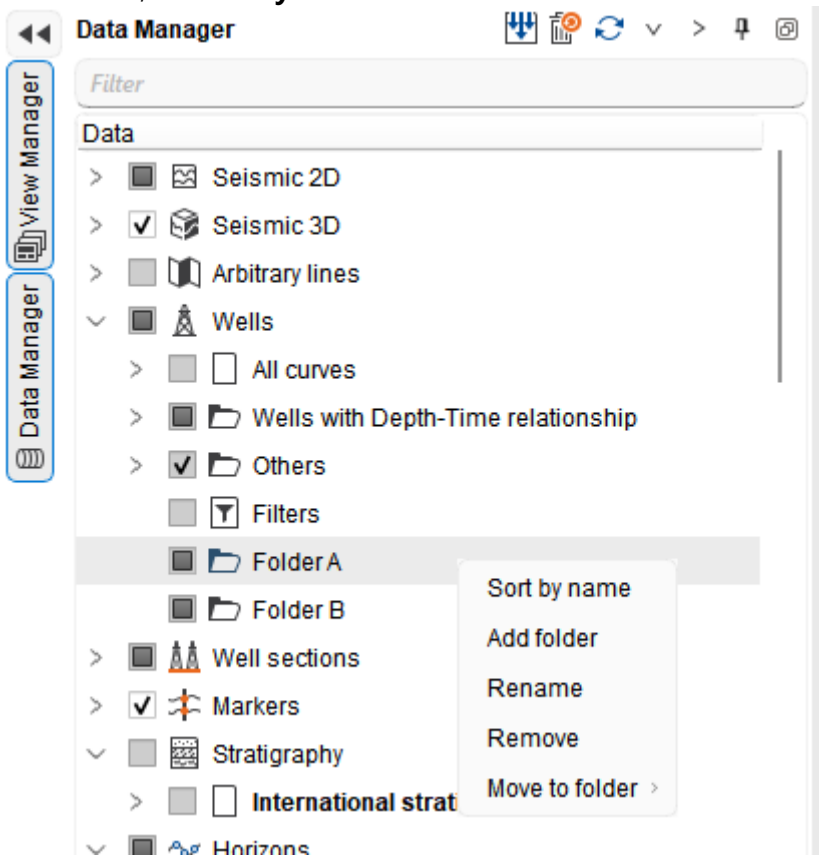
Both buttons act on the active view only – with a 3D view active they affect that 3D view, with a **Location map** active that Location map. They change visibility only: the objects, their data, their availability and the folder structure are not touched, and objects that cannot be visualized in

© 2026 Geomage 2003 LTD

the active window type (gray icon) are not affected. The Data Manager checkboxes and the **Visual Settings** panel are updated immediately.
Use the **Filter** field at the top of the panel to quickly find objects or folders by name and simplify navigation in large projects.

Sorting objects by name

Right-click a top-level data category and choose **Sort by name** to arrange its objects by name. Sorting ignores case and compares runs of digits by value, so numbered names line up as expected: 9-AX-2 comes before 12-AX-11. This option is available for the **Wells**, **Horizons**, **Maps**, **Seismic 2D**, **Seismic 3D**, **Wavelets** and **Multi-Z surfaces** categories. Sub-folders are listed before individual objects; in the Wells category the **All curves** and **Filters** branches remain pinned at the top.
In the **Wells** category, **Sort by name** is also available on every well folder and sub-folder: right-click the folder and choose it at the top of the folder command block, which continues with **Add folder**, **Rename**, **Remove** (offered only for an empty folder) and **Move to folder**. The command rearranges only the direct contents of that folder, so the other folders and the rest of the tree keep their order, and each sub-folder is sorted separately when the command is run on it. The folders that group LAS files and curves inside an individual well have no folder commands, so **Sort by name** is not offered there.



© 2026 Geomage 2003 LTD

Arranging objects manually

Objects can also be arranged by hand. Drag an object or folder and drop it at another position inside its own data category, or drop it on a folder of that category to move it into that folder. An object cannot be dragged from one data category into another.
In the **Wells** category the **All curves** and **Filters** branches always stay at the top of the list: they cannot be dragged, and nothing can be dropped above them or on the **Wells** category row itself. Drop wells and well folders below **Filters**.
A folder added with **Add folder** appears at the end of the list it is created in, and the objects already present keep their positions.

Data categories

Data manager contains predefined categories for specific data types:

Seismic 2D - contains any type of 2D seismic data such as post-stack, pre-stack, velocity, attributes etc.
Right mouse button on the icon allows to:

- Add new line
- Convert ASCII data to lines
- Clear lines
- Export all lines to CPS-3 line file
- Export all time lines to depth
- Export all time lines with shifts
- Receive lines
- Commit lines
- Add folder

Seismic 3D - contains any type of seismic 3D data such as post-stack, pre-stack, velocity, attributes etc.
Right mouse button on the icon allows to:

- Add new volume
- Convert ASCII data to volume
- Clear volumes
- Clear 3D visualization cache
- Receive volumes
- Commit volumes
- Add folder

Sections - each 3D survey (volume geometry) has a **Sections** folder that stores individual inline (IL) and crossline (XL) sections of that survey. The folder always contains two automatic entries, **Selected Inline <number>** and **Selected Crossline <number>**, whose numbers follow the currently selected inline and crossline; while nothing of that survey is selected they show its centre lines. These two entries have no context menu commands and do not react to double-

© 2026 Geomage 2003 LTD

click, but each of them has a working checkbox: select or clear it to show or hide that line on the Location map. Sections added with **Add section...** are listed as **IL <number>** and **XL <number>**.
Right mouse button on the **Sections** folder allows to:
- Add section... - choose the axis (Inline or Crossline) and the line number to add. The number defaults to the central line of the survey.

Right mouse button on an individual added section (**IL <number>** or **XL <number>**) allows to:
- Visualize - display that inline or crossline in the seismic views
- Remove

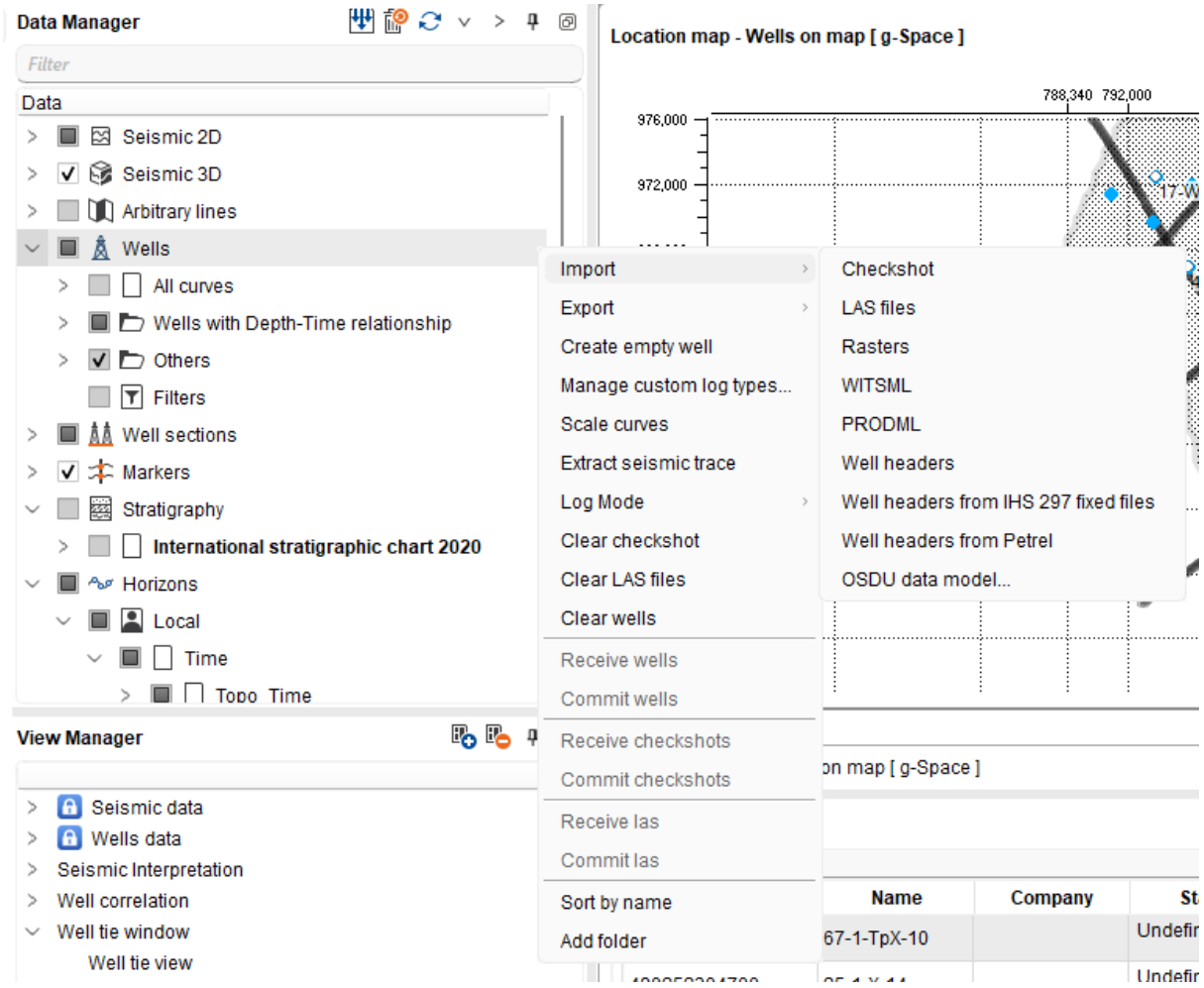
Double-click an added section to visualize it.

Arbitrary lines - contains all seismic arbitrary lines created in g-Space
Right mouse button on the icon allows to:
- Add arbitrary line
- Clear arbitrary lines
- Add folder

Wells - contains any type of well data
Right mouse button on the icon allows to:
- Import > Checkshot
- Import > LAS files
- Import > Rasters
- Import > WITSML
- Import > PRODML
- Import > Well headers
- Import > Well headers from IHS 297 fixed files
- Import > Well headers from Petrel
- Import > OSDU data model...
- Export > Wellheads
- Export > Inclinometry
- Export > Checkshot
- Export > LAS
- Create empty well
- Manage custom log types...
- Scale curves
- Extract seismic trace
- Log Mode > Curves or LAS files
- Clear checkshot
- Clear LAS files
- Clear wells
- Receive wells
- Commit wells

© 2026 Geomage 2003 LTD

- Receive checkshots
- Commit checkshots
- Receive las
- Commit las
- Sort by name
- Add folder



Well sections - contains all well sections created in g-Space
Right mouse button on the icon allows to:

- Add well section
- Clear well sections
- Add folder

Markers - contains well marker data with other attributes
Right mouse button on the icon allows to:

- Add marker set
- Clear
- Receive markers

© 2026 Geomage 2003 LTD

- Commit markers
- Add folder

Stratigraphy - contains the stratigraphic data
Right mouse button on the icon allows to:
- Load from XML
- Load default
- Clear
- Add folder

Horizons - contains seismic picking
Right-click the **Horizons** category to:
- Add new horizon
- Clear horizons

Right-click the **Time** or **Depth** node and choose **Create thickness maps**. On the **Depth** node, you can also choose **Create isopach maps**. These commands are available when the selected domain contains at least two horizons.
- Receive horizons
- Commit horizons

Faults - contains fault picking
Right mouse button on the icon allows to:
- Add new fault
- Clear faults
- Set meshing mode to Sticks or Surfaces
- Export faults
- Receive faults
- Commit faults

Topography - contains internal (from seismic headers) and external (from other sources) topography
Right mouse button on the icon allows to:
- Add folder

Raster files - contains all image files loaded in g-Space
Right mouse button on the icon allows to:
- Add raster files
- Add folder

Shapefile layers - contains loaded shapes and cultural data
Right mouse button on the icon allows to:
- Add shape files
- Load cultural points from text file

© 2026 Geomage 2003 LTD

- Load custom points from text file
- Add folder

Maps - contains all grids loaded externally and generated by running velocity model.
Maps are separated by types:
- Depth maps
- Velocity maps
- Calculated maps
- Time maps
- Attribute maps
- Imported maps
- Spectral decomposition maps
- Property maps
- Isochore maps
- Isopach maps
- Isochron maps

Right mouse button on the icon allows to:
- Clear maps
- Add folder

Right mouse button on **type folder** allows to:
- Remove
- Load from text file
- Load from CPS-3 grid file
- Load from ZMap grid file

Velocity model - contains models created in g-Space
Right mouse button on the icon allows to:
- Add velocity model
- Import velocity model
- Clear
- Add folder

Polygons - contains polygons created in g-Space
Right-click the **Polygons** category to:
- Add polygon
- Import > Import polygons from CPS-3 line file
- Import > Import polygons from DXF file
- Export > Export all polygons to CPS-3 line file
- Export > Export all polygons to shapefile
- Export > Export all polygons to DXF file
- Clear

© 2026 Geomage 2003 LTD

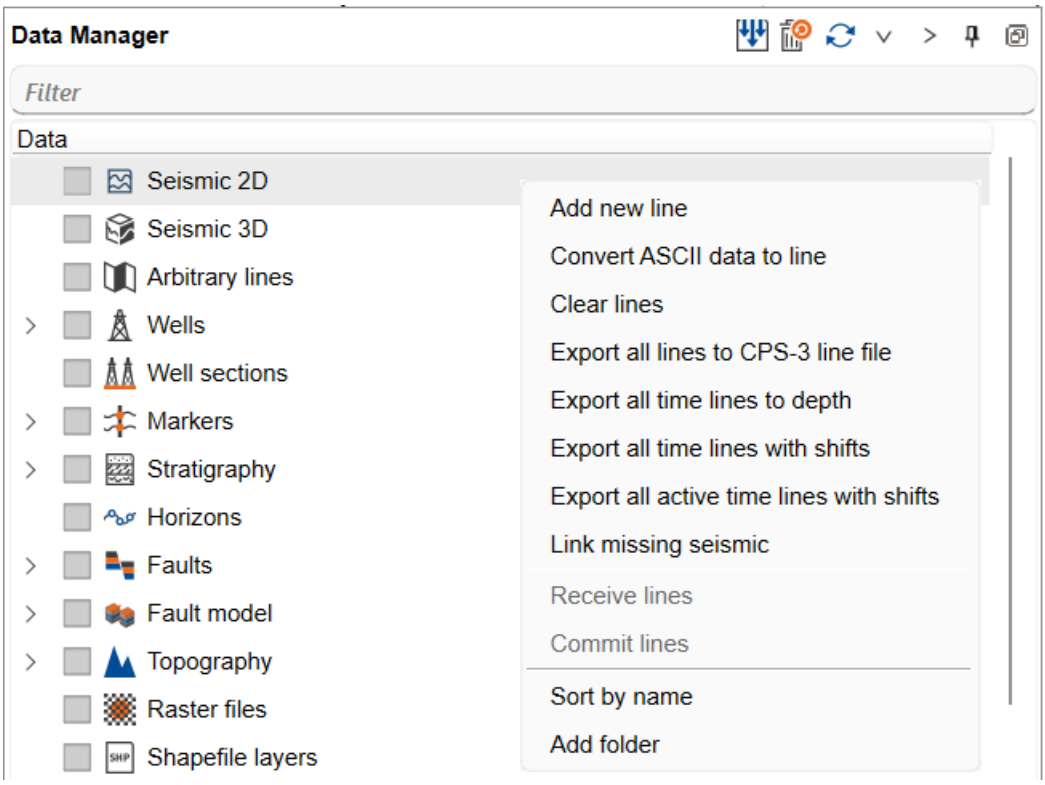
- Receive polygons
- Commit polygons
- Add folder

3D Grids - contains 3D grids
Right mouse button on the icon allows to:
- Create new grid
- Clear
- Add folder

Geo-bodies - contains created geobodies
Right mouse button on the icon allows to:
- Load from file
- Clear
- Add folder

3.2.1.1 Seismic 2D

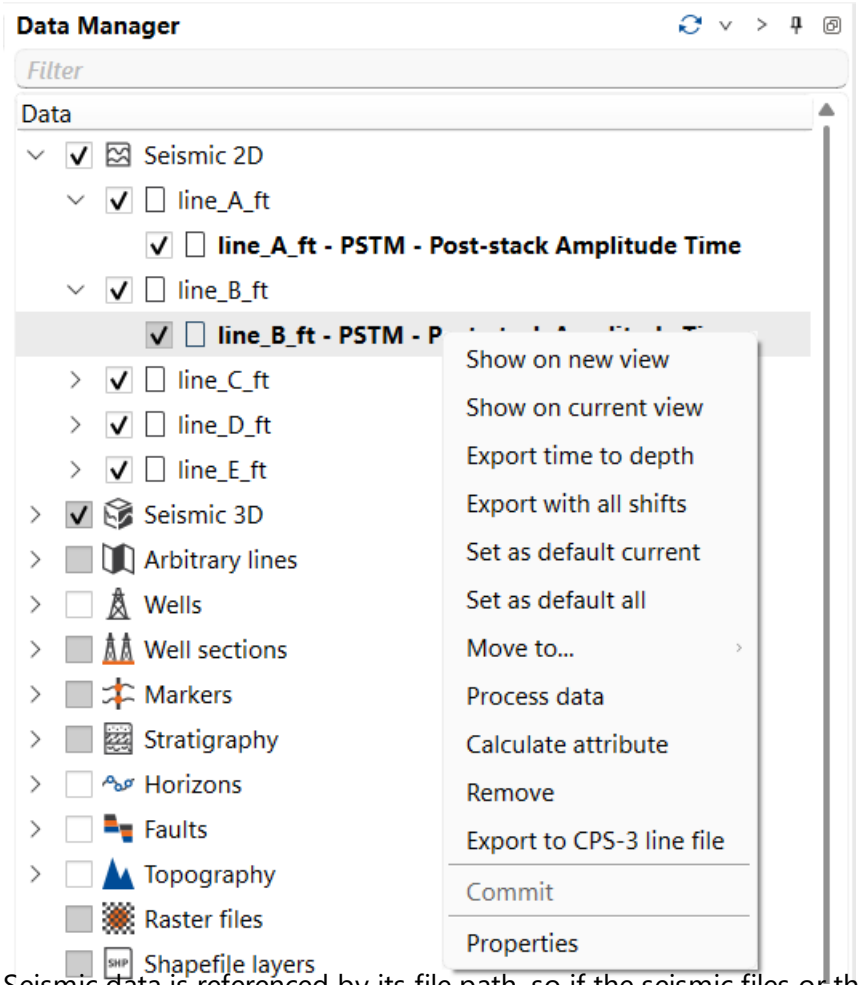
Seismic 2D - contains any type of 2D seismic data such as post-stack, pre-stack, velocity, attributes etc.



Right mouse button on the icon allows to:
- Add new line
- Convert ASCII data to line

© 2026 Geomage 2003 LTD

- Clear lines
- Export all lines to CPS-3 line file
- Export all time lines to depth
- Export all time lines with shifts
- Export all active time lines with shifts
- Link missing seismic
- Sort by name
- Add folder
- Receive lines
- Commit lines



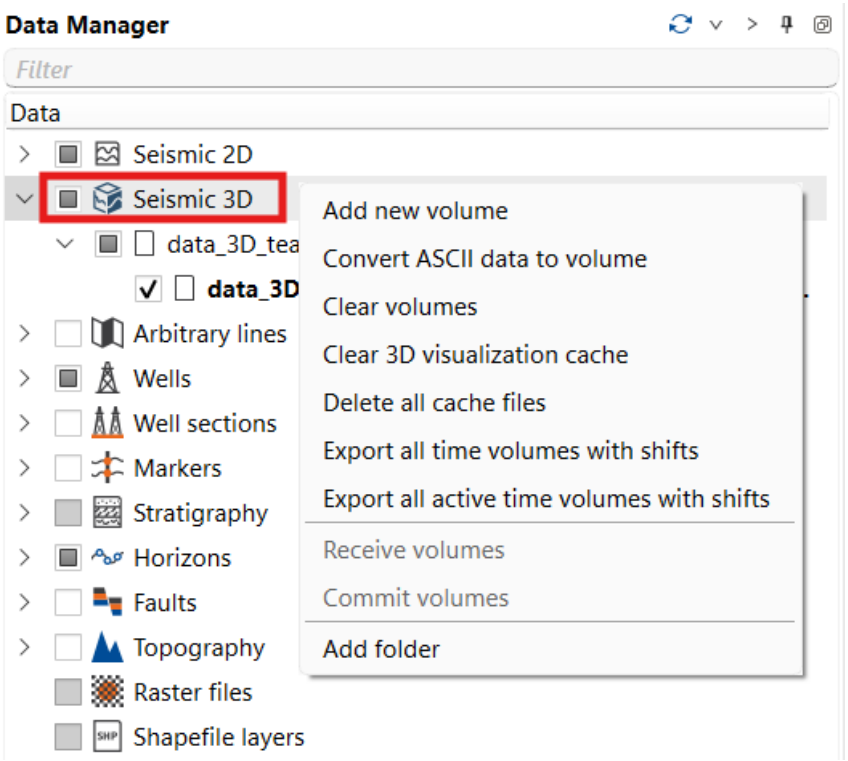
Seismic data is referenced by its file path, so if the seismic files or the project itself have been moved, g-Space can no longer find the files that were loaded before. Lines whose files are not found are kept in the project when it opens, and you can restore their locations at any time: right-click **Seismic 2D** and select **Link missing seismic**. In the **Select folder to search for missing seismic files** dialog, choose the folder that now holds the files. **Include subfolders** is cleared by default, so only the selected folder is searched; select it to search the whole folder tree below it as well.

© 2026 Geomage 2003 LTD

g-Space looks only for the seismic files that are currently missing and matches them by file name, so seismic that is still linked correctly is not changed. Every file that is found is relinked to its new location and can be used again, and a message reports how many missing files were found, how many were relinked and how many are still missing. The same action is available on **Seismic 3D**, and one run relinks 2D lines, 3D volumes and pre-stack gathers.

3.2.1.2 Seismic 3D

Seismic 3D - contains any type of seismic 3D data such as post-stack, pre-stack, velocity, attributes etc.



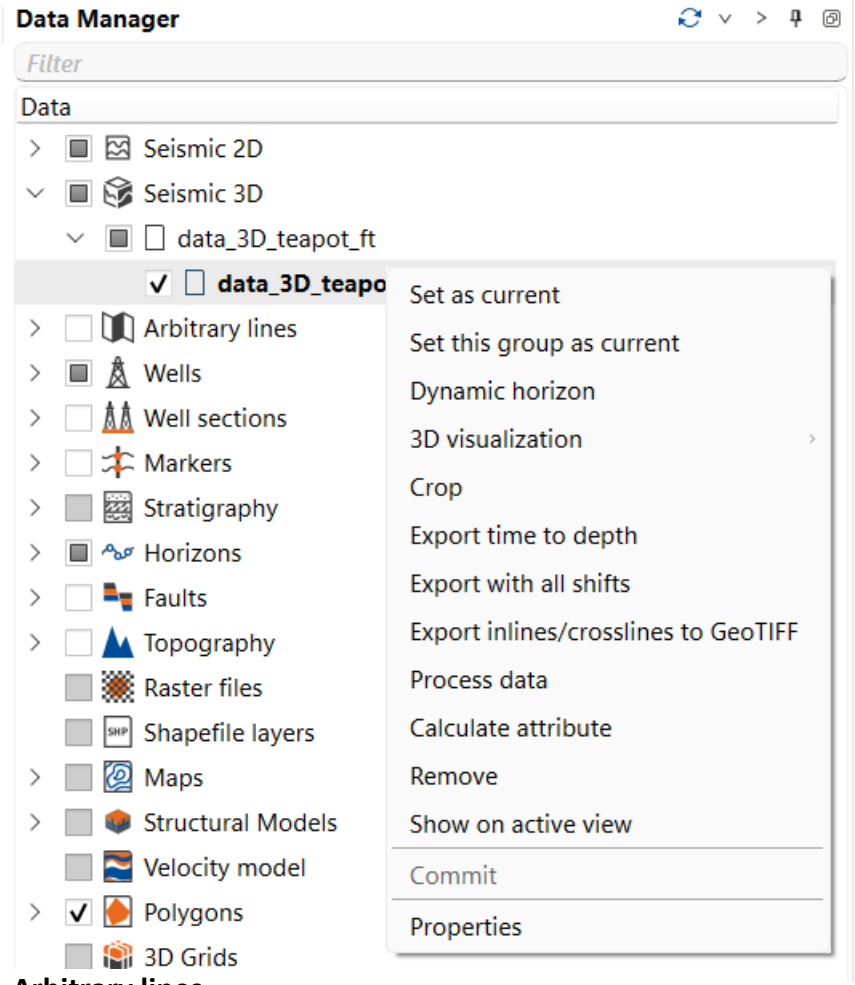
Right mouse button on the icon allows to:

- Add new volume
- Convert ASCII data to volume
- Clear volumes
- Clear 3D visualization cache
- Delete all cache files
- Export all time volumes with shifts
- Export all active time volumes with shifts
- Sort by name
- Add folder

© 2026 Geomage 2003 LTD

- Receive volumes
- Commit volumes

To crop a seismic volume, right-click it under **Seismic 3D** and select **Crop**. Set the output file and crop parameters, then select **Save to workflow** to add a **Crop 3D seismic volume** task to the current workflow without running it; running the workflow later performs the crop with those parameters. Select **Run** to crop the volume immediately. When you open the saved task from Workflows 535, its parameters are restored; select **Save** to update the task.



3.2.1.3 Arbitrary lines

Arbitrary lines - contains all seismic arbitrary lines created in g-Space
Right mouse button on the icon allows to:

- Add arbitrary line
- Clear arbitrary lines
- Add folder

An **arbitrary line** is a user-drawn path through a 3D seismic cube that extracts seismic traces along any geometry — not only the native in-line / cross-line grid. Use it to follow a geological

© 2026 Geomage 2003 LTD

feature, connect key wells on a single panel, or build a custom cross-section through a field. The path is defined interactively in a 3D or map view and the resulting gathers appear in a dedicated 2D view.

Creating an arbitrary line

1. Open a 3D view 244 or Location map 235 showing the target cube.
2. Enable arbitrary-line creation from the view toolbar and click the cube to place successive anchor points (**corners**).
3. Double-click the line to build it, or use **Update current arbitrary line** from the right-click menu to rebuild it at any time. g-Space builds the bin sequence along the polyline and extracts traces for every domain available on the cube (time, depth, offset …).
4. Manage existing lines (rename / delete / activate) from the Data manager 124 under the Arbitrary lines node.

While you are picking a line through a 3D cube or a free-hand 3D line, you can also reshape it. Press the left mouse button on or very close to an existing point to drag that point to a new position, or on the line between two points to drag the whole path; the line and the section extracted along it follow the points while the button is held, and the new geometry is kept when you release the button. Click clear of the line when you want to add another point instead. A line that follows 2D seismic lines cannot be reshaped this way.

When you add an arbitrary line, g-Space suggests a name in the **Set name of the arbitrary line** dialog: `New arbitrary line`, or `New arbitrary line 2`, `New arbitrary line 3` and so on when a line with the previous name already exists. Accept the suggestion or type your own name in **Arbitrary line name**; names must be unique, so g-Space asks again if the name you enter is already taken.

When you place the first point of a new line, g-Space appends a type suffix to its default name (`New arbitrary line`) so the different arbitrary-line types are easy to tell apart in the Data manager:

- `_2D` — a line that follows existing 2D seismic lines
- `_3D` — a line extracted through a 3D cube
- `_Free` — a free-hand 3D line

Only lines that still carry a suggested default name — with or without its counter — are renamed automatically, a line you have already renamed yourself keeps its name.

Parameters

These settings are grouped under **Arbitrary line params**:

© 2026 Geomage 2003 LTD

- **Create arbitrary line** — builds the line from the points you picked. When it is cleared, the picked points are discarded instead of producing a line. Enabled by default.
- **Arbitrary line step** — distance between consecutive bins along the line. Default `25 m`.
- **Arbitrary line max search distance** — maximum lateral distance from the polyline beyond which 3D traces are ignored when interpolating. Default `200 m`.
- **Arbitrary line moving step** — distance the whole line is shifted sideways each time you press **Page Up** or **Page Down** in the view. The line moves perpendicular to the straight line joining its first and last points, and the two keys move it in opposite directions. This works for a line through a 3D cube and for a free-hand 3D line that has at least two points, but not for a line that follows 2D seismic lines. Default `200 m`.
- **Straighten on 3D view** — when enabled, the line is rendered as a straight plane inside the 3D view for easier inspection.

What you can do with an arbitrary line

- Display extracted gathers in the dedicated 2D view and pick horizons / faults on them as on any seismic section.
- Inspect the nearest wells and nearest 2D lines to each corner — g-Space keeps per-corner associations for quick context switching.
- See which 2D seismic lines the path crosses — the view prints their names in the annotation area above the section. Clear **Show line names** in the View 208 section of the **Visual Settings** panel to hide these labels; the option is selected by default.
- Convert the extracted path into a standalone 2D line or export it to SEG-Y for sharing.

See Also

- 3D view 244
- Seismic view 240
- Seismic picking 586

3.2.1.4 Wells

Wells - contains any type of well data
A Wells folder can hold well headers, inclinometry, logs (LAS files / curves), markers, checkshots, comments and rasters/images.
Right-click the **Wells** icon in the **Data Manager** to access:

- Import (Checkshot, LAS files, Rasters, WITSML, PRODML, Well headers, Well headers from IHS 297 fixed files, Well headers from Petrel, OSDU data model...)
- Export (Wellheads, Inclinometry, Checkshot, LAS)
- Create empty well
- Manage custom log types...
- Scale curves

© 2026 Geomage 2003 LTD

- Extract seismic trace 147 (for all wells in the folder)
- Log Mode (Curves / LAS files)
- Clear checkshot
- Clear LAS files
- Clear wells
- Sort by name
- Add folder

Wells can be grouped in folders under **Wells**; use **Add folder** to create one. Right-click a well folder to run Extract seismic trace 147 for every well below it, wells in nested folders included, to open its **Export** submenu, or to **Remove** those wells. On the **Wells** folder itself, **Extract seismic trace** is unavailable until the project contains at least one well.

When the project is connected to a database, additional actions are available according to your permissions:

- Receive wells
- Commit wells
- Receive checkshots
- Commit checkshots
- Receive las
- Commit las

Deleting well data

Commands that delete well data ask you to confirm first. **Clear checkshot**, **Clear LAS files** and **Clear wells** on the **Wells** folder, and **Remove**, **Clear markers**, **Clear LAS info** and **Clear checkshots** on an individual well, open a **Confirm** message that names the data to be removed. Click **Yes** to delete it or **No** to keep the data; **No** is preselected. The same confirmation appears when you remove several selected wells, LAS files or log curves at once, and for the **Clear wells inclinometry** and **Clear all wells comments** commands on the Wells bar 46.

Log Mode

Log Mode controls how the logs of every well are arranged in the **Data Manager**. Right-click the **Wells** folder, point to **Log Mode** and choose **Curves** or **LAS files**; the mode currently in use is checked. The same setting is available from the **Log Mode** drop-down on the Wells bar 46.

Curves is the default mode and the arrangement the rest of this topic assumes: under each well, **Las** and the user item below it list the log curves under the names you gave them, curves that belong to a LAS set are grouped in a folder named after that set, and the **Wells** branch starts with the global **All curves** section.

In **LAS files** mode, each imported LAS file appears as its own item under **Las** and the user item, named after the file without its extension, and the curves it contains are listed inside it. Those curves are labelled with the original mnemonic read from the LAS file rather than the name you gave them, so a curve you renamed to Gamma Ray 1 is listed as GR. The index curve of the file comes first as **MD**, the remaining curves follow in alphabetical order, and two extra columns to

© 2026 Geomage 2003 LTD

the right of each curve show its type and its units. LAS set folders and the global **All curves** section are not shown in this mode.
Switching the mode only re-arranges how the logs are listed - no curve, name, type or file is changed - so you can switch back and forth freely.

Curve colours

You can give each log its own colour in the **Data Manager**. Right-click **All curves**, or one of the LAS set folders under it, and choose **Colorize** to give every curve name in that scope a distinct colour. Right-click a single curve under **All curves** and choose **Set color** to pick its colour yourself, or **Colorize** to have one assigned automatically. You can also select several curves - under **All curves** or under a well's **Las** list - and use **Colorize** on all of them at once. A single curve listed under a well has no colour command of its own.
The colour is stored with the project against the curve name, so the same log name keeps that colour in every well, and it is shown as the background of the curve icon in the **Data Manager**. In well log views the colour is used as the default colour of that curve, and views that are already open are redrawn. A colour saved in a well template 544 and a colour set for the curve in the view's **Curve Settings** both take precedence over it.

Displaying LAS curves

With a well log view active, select a log curve's checkbox in the **Data Manager** to add it to the active track. If the well has no active track, a new track is created. Clear the checkbox in the **Data Manager** to remove the curve from every track in the well log view. Selecting the checkbox again returns the curve to the track or tracks it was removed from - to several tracks at once when it was displayed on more than one - but a track deleted in the meantime is not recreated, and when none of the remembered tracks is left the curve is added to the active track as described above. The placement is remembered while this view stays open and is not stored in the project. This works for curves under an individual well and, in **Curves** mode, for the global logs under **All curves**.
To change one track only, right-click the track, choose **Advanced Track Settings...**, and select or clear the curve in the **Curves** section. This does not add or remove the same curve on other tracks.
With a time- or depth-domain 3D view 244 active, select a LAS curve's checkbox in the **Data Manager** to display the curve along its wellbore; clear the checkbox to hide it. A synthetic trace saved as a LAS curve can be displayed in the same way.
To change its 3D appearance, open Visual Settings 206, select the curve under **Las curves 3D** on the **Object(s)** tab, and adjust **Curve style**, **Line color**, **Line size**, and **Gain**.

Displaying checkshot curves

Every checkshot loaded on a well is listed under **Checkshot** in the **Data Manager**, and each checkshot holds two curve items, **TWT** and **Vint**, with the same checkbox as any other object. Only the checkshot currently in use for the well - the one shown in bold - supplies the curves that can be displayed.

© 2026 Geomage 2003 LTD

With a well log view active, select **TWT** or **Vint** to add that curve to the active track; if the well has no active track, a new track is created. Clear the checkbox to remove the curve from the view.
The same checkboxes also show and hide these curves along the wells drawn on inline, crossline and arbitrary line seismic sections, in time and in depth, where the display additionally follows the **Show las curves on seismic views** property described in Module Properties 197. In a 3D view only LAS curves are drawn along the wellbore.

Adding a comment to a well

Right-click an individual well in the Data Manager and choose **Add comment** to attach a depth-located note to that well. The dialog is titled with the selected well name and asks for an **MD** and the **Comment** text. The MD must lie within the well's trajectory - between its top and bottom MD - so values above the well top or below the well bottom are rejected, and empty comment text is not accepted. You can add several comments to the same well at different MD values; they are then available in well-related displays such as the Well sections 150 and Well correlation view 296.

See Also

- Wells bar 46
- Wells data workspace 232

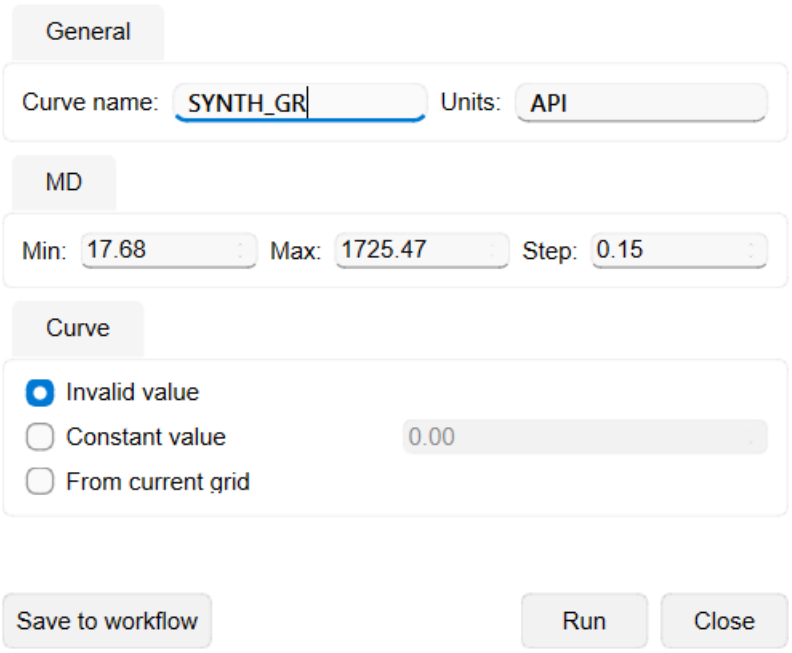
3.2.1.4.1 Create curve

The **Create a curve** wizard adds a new, synthetic LAS curve to an existing well. Use it to create placeholder curves before a calculation, or to seed a curve with a constant value or with values read from the current grid without running an external editor.

Launching the wizard

Open Data manager 124, right-click the target well and choose **Create a curve**. If the well already contains log data, the **MD** boxes open pre-filled with the start, stop and step of its first LAS file.

© 2026 Geomage 2003 LTD



Input Parameters

- **Curve name** — identifier used to list the curve in the Data manager and referenced by downstream tools. The well must not already contain a curve with this name.
- **Units** — the unit label stored with the curve.
- **MD** — **Min**, **Max** and **Step** set the measured-depth range and the sampling interval of the new curve. **Max** must be greater than **Min**, and **Step** must be positive.
- **Curve** — how the samples are filled. **Invalid value** marks every sample as invalid, so the curve is created empty and can be filled in later. **Constant value** writes the number entered beside the option into every sample. **From current grid** takes each sample from the current grid property at the point where the well trajectory crosses that depth, so a current grid and a current grid property must be set.

After the curve is created, populate it with data via the LAS editing 320 dialog or any other petrophysical wizard that writes output curves.

Running and saving to Workflow

The buttons along the bottom of the wizard are:

- **Run** — creates the curve now with the current settings.
- **Save to workflow** — adds the current settings, including the selected well, to the active workflow 535 as a **Create a curve** job instead of creating the curve now. The curve is created when the workflow is executed.

© 2026 Geomage 2003 LTD

- **Close** — closes the wizard without creating the curve and without saving the settings.

Opening a saved job from the Workflows 535 wizard reopens **Create a curve** with all of its stored settings restored, so they can be reviewed, edited and saved back to the same job. In that mode the wizard shows **Save** and **Cancel** instead.

Load curve from ASCII

To load an existing curve from a text file instead of creating an empty one, right-click the well in the Data manager 124 and choose **Load curve from ASCII**. This opens a text-import wizard titled **Import log [well name]** where you map the file columns to the **MD**, **Value**, **Mnemonic** and **Units** fields to load the curve from a `.txt` / ASCII file.

See Also

- LAS editing 320
- Well log calculator 558
- Import LAS files 407
- Data manager 124
- Workflows 535

3.2.1.4.2 Load raster

The **Open raster** wizard imports a well-log raster image (a scanned or digital strip log) into a well. It displays the image in an interactive crop view so the user can clip away headers, borders and margins before the raster is attached to the well.

Launching the wizard

In Data manager 124 right-click the target well and choose **Load raster**. This opens the **Open raster** wizard for that well. For batch loading of rasters described by XML metadata use the Open rasters 161 wizard instead.

Input Parameters

- **Image file** — the raster to load (PNG, JPEG, TIFF or BMP).
- **Crop rectangle** — eight handles (four corners + four edges) let the user trim the image on top, bottom and sides. Drag any handle to resize; drag inside the rectangle to move it as a whole.

© 2026 Geomage 2003 LTD

Depth referencing

After cropping, the image is positioned along the well axis by setting the depth interval it covers — the **MD begin** and **MD end** values — together with the display parameters. These values tie the top and bottom of the raster to measured depth so the image lines up with the curves, markers and comments of the well.

For batch loading of several rasters described by XML metadata (raster filename, UWI, well name, depth ranges and calibration), use the separate Open rasters 161 wizard instead.

See Also

- Open rasters 161
- Data manager 124

3.2.1.4.3 Scale Curves

Scale Curves is a tool for upscaling (blocking) LAS well-log curves along the well trajectory. It resamples a high-resolution log into coarser cells - either following the layering of an existing 3D grid or using a fixed scale interval you define - and writes the result back as a new curve on the selected well(s). The upscaled curves are typically used to condition property modeling and to compare log data with grid-cell resolution.

Purpose

The tool is designed for:
- Blocking (upscaling) LAS curves from sample resolution to grid-cell or interval resolution
- Preparing well-log data for use in property modeling and geostatistics
- Averaging a log over regular depth intervals along the borehole path
- Reconciling log resolution with the vertical cell size of a 3D grid

Requirements

- At least one well with loaded log curves (**LAS files / curves**) in the Wells 137 folder of the Data Manager 124
- Valid deviation / inclinometry data on the well when the **TVD** sampling direction is used
- For the **Using a Grid** method - at least one 3D grid available in the project

© 2026 Geomage 2003 LTD

Launching the Wizard

In the Data Manager 124, right-click the Wells 137 folder and choose **Scale curves** from the context menu. The **Scale curves** dialog opens.

The dialog is split into two areas: a controls panel on the left (well, curve, depth interval, scaling method and output options) and an **Input/Output** preview panel on the right that plots the source curve against the scaled result.

Step-by-Step Guide

Configure the controls in the left panel from top to bottom, then generate the scaled curve.

Selecting the Well and Curve

- **Well:** choose the well whose log will be upscaled.
- **Batch** - enable this checkbox and use the adjacent dropdown to apply the same settings to a group of wells (for example, all wells) in a single run, instead of processing one well at a time.
- **Curve:** select the log curve to upscale. The dropdown lists the curves available on the selected well.

Setting the Depth Interval

The depth-range widget limits the portion of the log that is processed. It offers **MD range**, **TVD range**, **TVDSS range** and **Markers range** tabs, each with **Start** and **End** fields. By default the full depth range of the selected well is used; leave the fields empty (Undefined) to keep the full range, or enter values to restrict processing to an interval.

Choosing the Scaling Method

The **Method** group offers two ways to define the cells that the curve is averaged into. Select one of the two radio buttons; the settings below the buttons change accordingly.

© 2026 Geomage 2003 LTD

Using a Grid (default)

With **Using a Grid** selected (the default method), the curve is upscaled along the well trajectory using the cell layering of an existing 3D grid. Each segment of the trajectory that passes through a grid cell defines one averaging interval, so the scaled curve matches the vertical resolution of that grid.

- **Grid:** choose the 3D grid whose cells drive the upscaling. The dropdown lists all grids available in the project.

Create New Scaling

With **Create New Scaling** selected, the curve is blocked into regular intervals of a size you define, independent of any grid. Use this method when no suitable grid exists or when you want a fixed sampling step.

- **Scale interval** (shown with the current depth unit, m or ft): the cell size of the new scaling - for example 5 or 10. The default is 10. The curve is averaged over successive intervals of this length along the trajectory.

Sampling Direction

The **Sampling direction** dropdown controls how depth intervals are measured when averaging the log:

- **Along trajectory** (default) - intervals are measured along the borehole path (measured depth), following the well deviation.
- **TVD** - intervals are measured in true vertical depth. This option is available only when the well has valid deviation / inclinometry data and the chosen depth range maps to a finite TVD range; otherwise it is disabled and the tool falls back to **Along trajectory**.

Averaging Method

The **Averaging** dropdown defines how the log samples inside each cell are combined into a single value:

- **Most Common** (default) - the most frequent value; suited to categorical / discrete logs (for example facies).
- **Average** - the arithmetic mean of the samples.
- **RMS** - the root-mean-square value.
- **Median** - the middle value, less sensitive to outliers.

© 2026 Geomage 2003 LTD

- **Minimum** - the lowest sample value in the cell.
- **Maximum** - the highest sample value in the cell.

Output Suffix and Preview

In the **Suffix** field, set the text appended to the source curve name to build the name of the new upscaled curve. The default suffix is `_new`; edit it as needed. If the suffix is left empty, the source curve is overwritten with the upscaled version and the tool asks you to confirm.
Press **Apply** to compute a preview: the source log appears in the **Input** plot and the blocked result in the **Output** plot of the **Input/Output** panel. Adjust the settings and re-apply until the result is satisfactory. **Apply** only previews - it does not save the curve.

Saving the Results

Press **Save** to write the upscaled curve. The new curve is named `<curve><suffix>` and is added back to the selected well(s) in the Data Manager 124. If a curve with the same name already exists, the tool asks whether to overwrite it. When **Batch** is enabled, the curve is generated for every well in the batch selection.
Use **Save to workflow** instead to store the current settings as a reusable step in the processing workflow rather than running the operation immediately. **Close** dismisses the dialog.

Results

- A new upscaled (blocked) curve, named after the source curve plus the **Suffix**, stored on each processed well.
- The curve is available in the Wells 137 folder and in the well-log displays of the **Well-data-workspace**.
- The upscaled curve can be used directly as input for property modeling and geostatistical workflows.

Recommendations

- Match the scale to your target grid: use **Using a Grid** when the upscaled logs will condition properties of that grid, so the log and cell resolutions agree.
- Choose the averaging method to fit the log type - use **Most Common** for categorical logs (facies, lithology) and **Average**, **RMS** or **Median** for continuous petrophysical logs.

© 2026 Geomage 2003 LTD

- Preview with **Apply** before saving to confirm the blocked curve honours the trend of the source log.
- Keep a non-empty **Suffix** to preserve the original high-resolution curve alongside the upscaled version.
- Use **TVD** sampling for strongly deviated wells when the target intervals are defined vertically; keep **Along trajectory** for measured-depth workflows.
- For the reverse operation - populating a well log from a 3D grid property - use the **Extract Grid Property to LAS** command available on a grid property in the Data Manager 124.

See Also

- Wells 137
- Wells data workspace 232
- Import LAS files 407
- Well log calculator 558
- Properties modeling 776
- Seismic Data Upscaling 782

3.2.1.4.4 Extract seismic trace

Extract seismic trace writes the seismic trace nearest to a well into a new LAS curve on that well, resampled along the wellbore at a step you choose. The result is an ordinary log curve, so the seismic amplitudes can be displayed and compared with the other logs of the well. The same settings can also be stored as a task in a project workflow 535 and re-run later. The command also works on a whole scope of wells: started on the **Wells** folder or on one of the well folders inside it, it extracts a trace for every well below that node in one run.

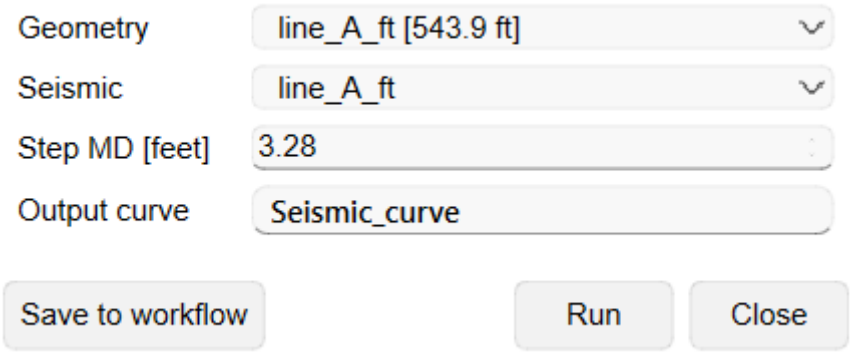
Requirements

- A well with a trajectory in the Wells 137 folder of the Data Manager 124.
- Seismic data lying within the **Nearest seis distance** search radius of the well head, set in the Module properties 197. Only the geometries and seismic data found inside that radius are offered, and the dialog does not open when nothing is found - increase the value when the nearest seismic lies further away.
- Checkshots on the well when the selected seismic is in the time domain, so that seismic times can be converted to depth along the well.

Extracting a trace for one well

© 2026 Geomage 2003 LTD

1. In the Data Manager 124, right-click a well under Wells 137 and choose **Extract seismic trace**.
2. Set the parameters described below.
3. Press **Run** to extract the trace now, or **Save to workflow** to keep the settings as a workflow task instead.



Parameters

- **Geometry** - the seismic geometry the trace is taken from. The list holds the geometries found near the well, each entry showing its distance from the well.
- **Seismic** - the seismic data set to sample. The offered list follows the selected geometry.
- **Step MD** (shown with the current depth unit) - the sampling interval of the resulting curve along measured depth.
- **Output curve** - the name of the curve written to the well. The default name is `Seismic_curve`; the name cannot be empty.

Buttons and workflow support

- **Save to workflow** - stores the selected well together with **Geometry**, **Seismic**, **Step MD** and **Output curve** as an **Extract seismic trace** task in the current workflow. Nothing is extracted at this point - the trace is produced when the workflow runs. The dialog stays open and keeps the entered values, so several tasks can be saved in turn.
- **Run** - extracts the trace immediately with the current settings and keeps the dialog open, so the settings can be adjusted and the extraction repeated. If the well already holds a curve with the output name, you are asked whether to overwrite it.
- **Close** - closes the dialog.

If the project contains no workflow yet, **Save to workflow** asks for a name and creates one. Reopening the task from the workflow restores the dialog with its saved settings so they can be reviewed and edited; **Save** writes the changes back to the task and **Cancel** closes the dialog.

© 2026 Geomage 2003 LTD

When the workflow runs, the trace is extracted with the stored settings, and an existing curve with the same name is replaced without a prompt.
The task refers to the geometry and the seismic data by name. If the saved geometry is no longer present in the project, the task reports it as unavailable and no curve is produced. See Workflows 535 for how workflows are built, ordered and executed.

Extracting traces for several wells

The same command is available on the Wells 137 folder itself and on any well folder inside it, including the folders you created. It then covers the whole scope below that node, wells in nested folders included, and extracts a trace for all of them in one run. Only 3D seismic volumes are offered for such a run, because the wells of a scope usually lie all over the survey.

1. In the Data Manager 124, right-click Wells 137 or one of the well folders under it and choose **Extract seismic trace**.
2. Choose the **3D seismic volume** to sample. The list holds the volumes of every 3D geometry that reaches at least one well of the scope within the **Nearest seis distance** radius; when the same volume name occurs in more than one geometry, that geometry is added to the entry in brackets. The dialog also states how many wells the selected scope holds.
3. Set **Step MD** and **Output curve** as for a single well, then press **OK** to start the extraction or **Cancel** to leave the wells unchanged.

Every well takes its trace from the part of the selected volume nearest to its own well head, so one run can cover wells spread over the entire volume. A well the volume does not reach within the search radius is skipped, and with a volume in the time domain a well without checkshots is skipped as well, because its times cannot be converted to depth. The remaining wells are processed one after another - a skipped or failing well does not stop the others - while a progress window shows how far the run has come and lets you stop it.
If a well already holds a curve with the output name, you are asked whether to replace it. **Yes** and **No** answer for that well only, **Yes to All** and **No to All** apply the same answer to the wells that follow, and **Abort** stops the run.
A summary closes the run. It reports for how many wells of the scope the curve was extracted and lists them by name, followed by the wells that were skipped - because the well has no time-depth relation, because the selected volume is out of reach, or because the curve already exists - the wells left unprocessed when the run was stopped, and the wells that failed, each with its reason. If the scope holds no well with a trajectory, if no 3D volume lies near it, or if none of its wells can be processed with the chosen volume, the command reports that and nothing is calculated; a volume you expect but do not find in the list usually lies further away than **Nearest seis distance** allows.
A batch run is always carried out immediately and cannot be kept as a workflow task - **Save to workflow** is offered for a single well only.

© 2026 Geomage 2003 LTD

Result

The extracted trace is added to the well as a LAS curve named after **Output curve** and appears under that well in the Data Manager 124. It behaves like any other LAS curve on the well - see Wells 137 for how log curves are displayed. If none of the well depths map onto the selected seismic - for example when the checkshots do not overlap the seismic, or **Step MD** is too large - the extraction reports this and no curve is created.

See Also

- Wells 137
- Workflows 535
- Well correlation view 296

3.2.1.5 Well sections

Well sections - contains well-log correlation sections (multi-well panels).
Right mouse button on the icon allows to:
- Add well section
- Clear well sections
- Remove template
- Add folder

Right mouse button on a well section additionally allows to:
- Set default
- Rename
- Remove
- Select template

Remove template is a submenu of saved section templates and appears only when at least one template exists.

See Also

- Well log correlation 538
- Setting Up a Well Template 544

3.2.1.6 Markers

Markers - contains well marker data with other attributes
Right mouse button on the icon allows to:
- Add marker set
- Clear
- Receive marker sets

© 2026 Geomage 2003 LTD

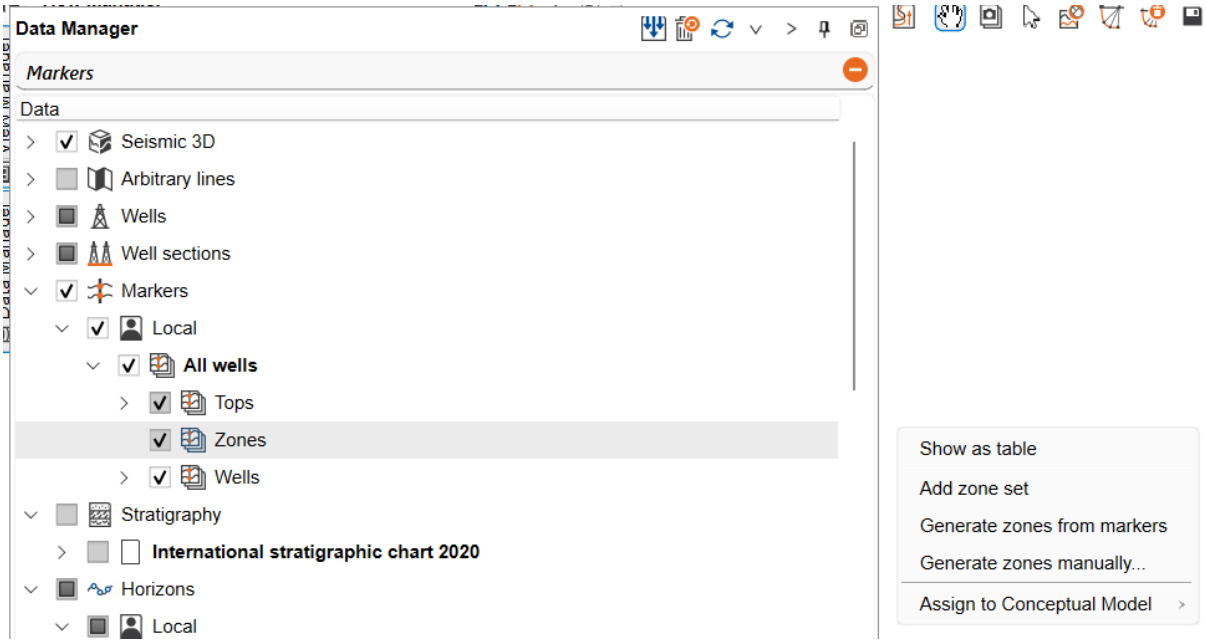
- Commit marker sets
- Add folder

Right mouse button on a marker set additionally allows to:

- Import markers struct
- Import markers ages
- Auto-create Zones
- Create Zones manually...

Showing marker data as tables

Right-click the **Tops** subsection, an individual top marker, the **Zones** subsection or an individual zone and choose **Show as table**. The command is the first menu item and opens the matching data in the Table view 285. A subsection shows all of its rows; an individual item filters the same table to that marker or zone.



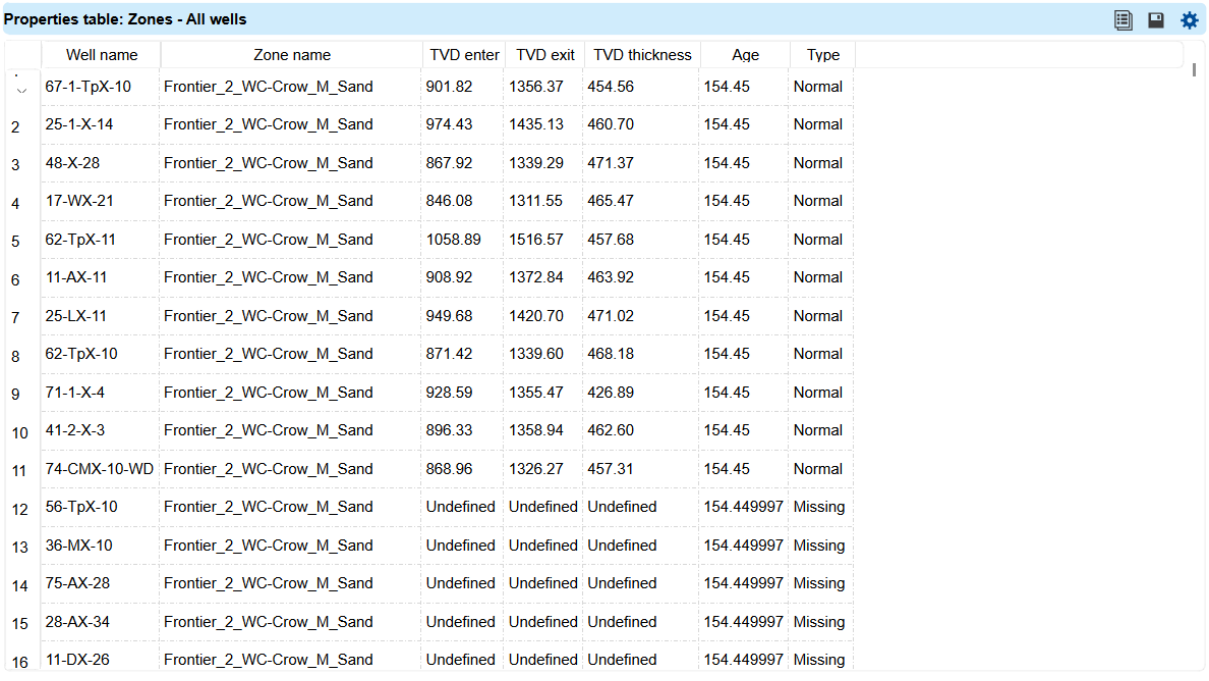
- **Tops** - the **Properties table: Marker tops - <marker set>** table contains one row for every picked marker in every well. Its columns are **Well name**, **Marker**, **MD**, **TVD**, **TVDSS**, **TWT (s)**, **X**, **Y**, **Age** and **Type**, followed by one column for each custom marker attribute defined in the set. **Type** is Marker, Fault or Intrusion. **TWT (s)** is Undefined when a well has no checkshot, and a custom attribute is Undefined where that marker does not carry it. Wells with no pick from the set are omitted. From an individual marker, the filtered table is titled **Marker top - <marker set> - <marker>**.
- **Zones** - the **Properties table: Zones - <marker set>** table contains one row per zone for every well in the project and includes every zone whose top or base refers to this marker dataset, across all zone sets. Its columns are **Well name**, **Zone name**, **TVD enter**, **TVD exit**, **TVD thickness**, **Age** and **Type**. **Age** is the average age of the top and base markers. **Type** is

© 2026 Geomage 2003 LTD

Normal, Inverted when the base is above the top, or Missing when either pick is absent. Inverted rows have an Undefined thickness; Missing rows have Undefined enter, exit and thickness values. From an individual zone, the filtered table is titled **Zone - <marker set> - <zone>**.

The **Marker tops** table is editable. **Marker**, **MD**, **Age**, **Type** and the custom attribute columns accept new values, while **Well name**, **TVD**, **TVDSS**, **TWT (s)**, **X** and **Y** are calculated from the pick and stay read-only on a grey background. **Marker**, **Age** and **Type** belong to the marker itself, so a new value there applies to that marker in every well of the set and a new name renames it everywhere, whereas a new **MD** or custom attribute value changes the edited row only. **Type** accepts only Marker, Fault or Intrusion. Edits are held in the table until **Save edits** writes them to the marker set; see Table view 285 for how held values are checked, saved or discarded.

The **Zones** table is read-only. Its values are derived from the marker tops the zones are built from, so edit those tops instead.



Properties table: Zones - All wells

	Well name	Zone name	TVD enter	TVD exit	TVD thickness	Age	Type
1	67-1-TpX-10	Frontier_2_WC-Crow_M_Sand	901.82	1356.37	454.56	154.45	Normal
2	25-1-X-14	Frontier_2_WC-Crow_M_Sand	974.43	1435.13	460.70	154.45	Normal
3	48-X-28	Frontier_2_WC-Crow_M_Sand	867.92	1339.29	471.37	154.45	Normal
4	17-WX-21	Frontier_2_WC-Crow_M_Sand	846.08	1311.55	465.47	154.45	Normal
5	62-TpX-11	Frontier_2_WC-Crow_M_Sand	1058.89	1516.57	457.68	154.45	Normal
6	11-AX-11	Frontier_2_WC-Crow_M_Sand	908.92	1372.84	463.92	154.45	Normal
7	25-LX-11	Frontier_2_WC-Crow_M_Sand	949.68	1420.70	471.02	154.45	Normal
8	62-TpX-10	Frontier_2_WC-Crow_M_Sand	871.42	1339.60	468.18	154.45	Normal
9	71-1-X-4	Frontier_2_WC-Crow_M_Sand	928.59	1355.47	426.89	154.45	Normal
10	41-2-X-3	Frontier_2_WC-Crow_M_Sand	896.33	1358.94	462.60	154.45	Normal
11	74-CMX-10-WD	Frontier_2_WC-Crow_M_Sand	868.96	1326.27	457.31	154.45	Normal
12	56-TpX-10	Frontier_2_WC-Crow_M_Sand	Undefined	Undefined	Undefined	154.449997	Missing
13	36-MX-10	Frontier_2_WC-Crow_M_Sand	Undefined	Undefined	Undefined	154.449997	Missing
14	75-AX-28	Frontier_2_WC-Crow_M_Sand	Undefined	Undefined	Undefined	154.449997	Missing
15	28-AX-34	Frontier_2_WC-Crow_M_Sand	Undefined	Undefined	Undefined	154.449997	Missing
16	11-DX-26	Frontier_2_WC-Crow_M_Sand	Undefined	Undefined	Undefined	154.449997	Missing

Creating zones from markers

A zone is the interval between two marker tops of the same marker set. **Auto-create Zones** and **Create Zones manually...** stay disabled until the marker set holds at least two tops. Zones created from a marker set are listed under it, in its **Zones** subsection.

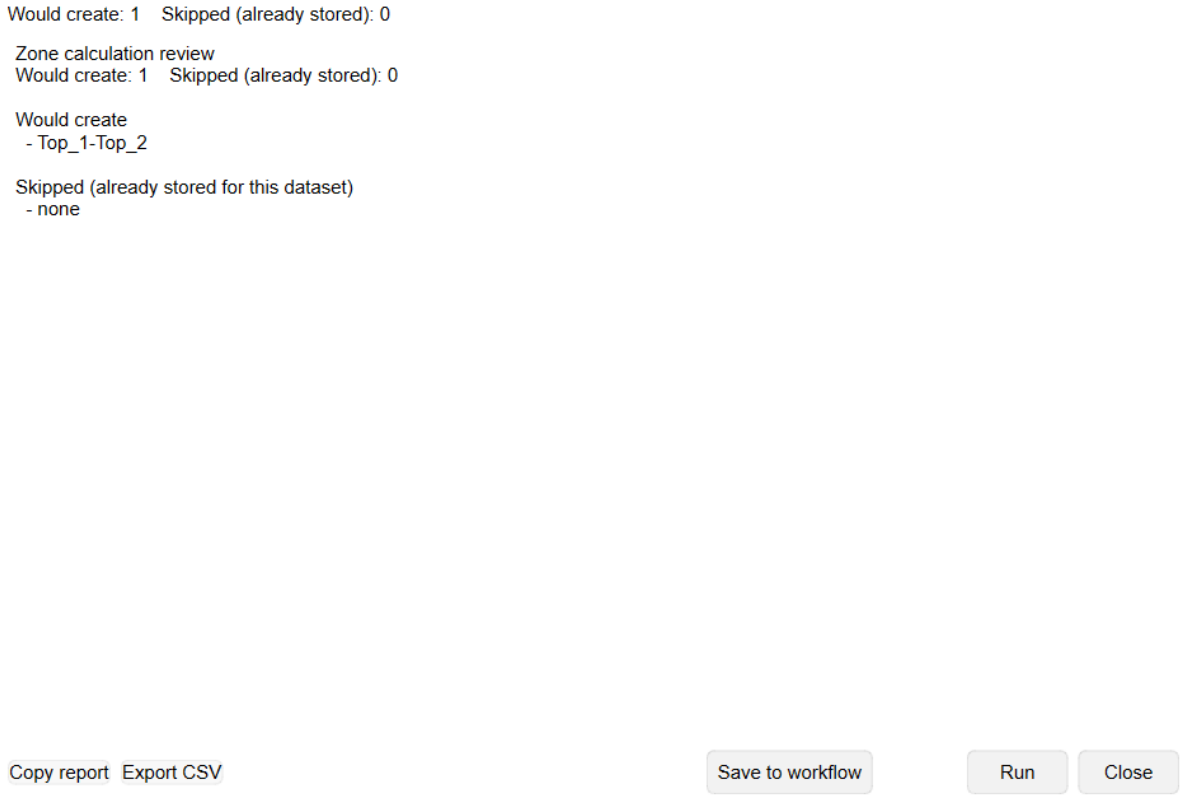
The **Zones** subsection offers the same two actions under different names, **Generate zones from markers** and **Generate zones manually...**, disabled under the same condition. Its menu

© 2026 Geomage 2003 LTD

also holds **Add zone set**, which asks for a name in the **Set name of the zone set** dialog, proposes **New zone set** and refuses a name that another zone set already carries. Zones belong to a zone set: new zones go to the current zone set, and when the project holds no zone set yet, g-Space creates one named **Default** and places them there.

Auto-create Zones pairs the tops in their current top-to-bottom order and opens the **Zone calculation review** dialog before anything is created. The report states how many zones would be created and which pairs are skipped because they are already stored for this marker set. Click **Run** to create them, **Save to workflow** to add the calculation to the current workflow instead, or **Close** to leave the data unchanged. **Copy report** and **Export CSV** keep the report for later use.

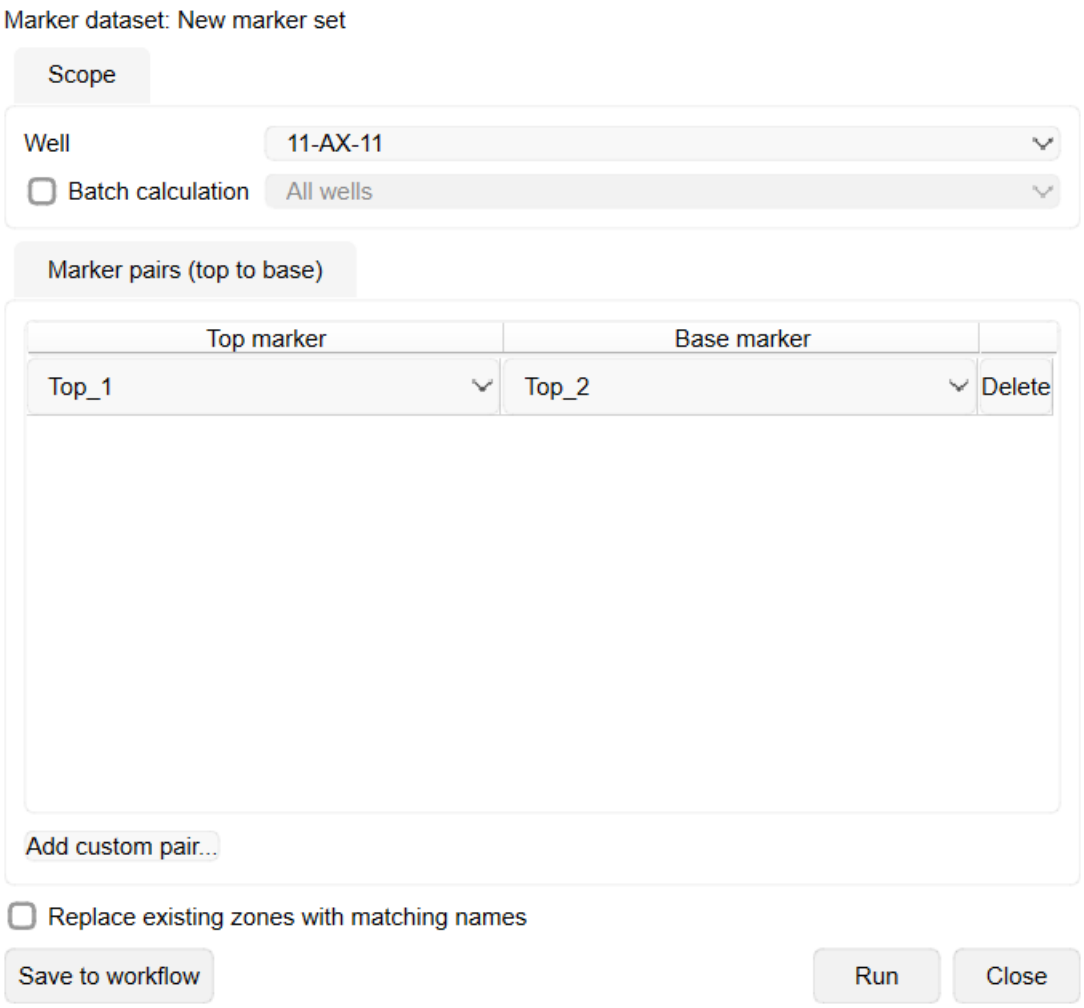
The workflow job saved from this report keeps the automatic pairing rule rather than the pairs listed in it. Each time the job runs it pairs the tops of the marker set in their current top-to-bottom order, covers every well in the project, skips the pairs already stored for that set and never replaces an existing zone, then opens the **Zone calculation review** dialog with the TVD thickness obtained for each zone and well. Running the job again after a top has been added therefore creates the zones that are missing, while running it when nothing is missing creates nothing and lists the pairs as skipped. The job is listed in the workflow as **Create zones from markers**.



© 2026 Geomage 2003 LTD

Create Zones manually... opens the **Create zones from markers** dialog, where the wells and the marker pairs are set before the zones are built:

- **Scope** - the wells the calculation covers: choose one **Well** by name, or select **Batch calculation** and pick a group of wells from the list beside it: **All wells** to process every well in the project, or a well filter, a Data Manager well folder, or a well section to limit the calculation to that group.
- **Marker pairs (top to base)** - the pairs derived from the order of the tops are listed with a **Top marker** and a **Base marker**. Change a pair in the drop-down lists, remove a row with **Delete**, or add one with **Add custom pair...**.
- **Replace existing zones with matching names** - overwrite zones that already carry the same name instead of adding a numbered copy.



Run creates the zones and then opens the **Zone calculation review** dialog with the TVD thickness obtained for each zone and well. The dialog summarises the run as **Full**, **Missing**, **Inverted** and **Skipped**, and lists the wells behind each of these counts. **Save to workflow** stores the same settings as a workflow job instead of running them now.

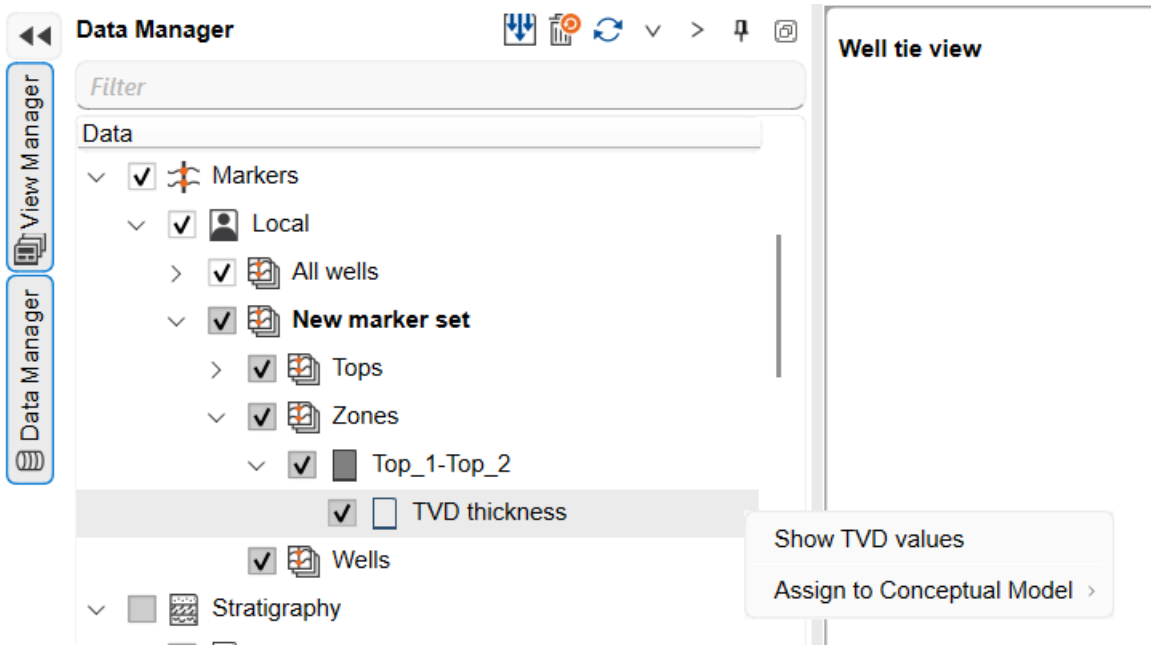
© 2026 Geomage 2003 LTD

A negative thickness is never stored. If a well has no pick for the top or the base marker, its TVD thickness is stored as None. If the base marker of a well resolves above its top marker, the zone is still created, but the TVD thickness for that well is also stored as None and the well is listed in the report under **Inverted intervals (stored as None)**. A negative TVD would indicate a fault repeat or a mis-pick, so these wells are left as None instead of passing a negative thickness on to the tools that use the zone.

Each created zone keeps its result as a **TVD thickness** item beneath it in the **Zones** subsection. To reopen the report for a single zone later, right-click that zone or its **TVD thickness** item and choose **Show TVD values**. The **Zone calculation review** dialog then lists every well of that zone with its TVD thickness and the TVD of its top and base marker; **Copy report** and **Export CSV** are available there as well.

A zone built from a pair of marker tops is named **<top marker>-<base marker>**, with a numbered suffix when the zone set already holds that name. The name follows the markers and cannot be edited: the menu of such a zone holds **Show as table**, **Show TVD values**, **Change color** and **Remove** but no **Rename**, and renaming its top or base marker under **Tops** renames the zone to match the new marker names. An attempt to rename it anyway is refused with a message explaining that zone names are derived from marker names and cannot be changed manually. Only a zone that is not built from a pair of marker tops can still be renamed.

Removing a marker that zones are built from removes those zones as well, so g-Space asks for confirmation first and lists the dependent zones as **<zone set>/<zone>**. Cancel keeps both the markers and the zones. The same confirmation appears when several markers, a marker group or the whole marker set is removed.



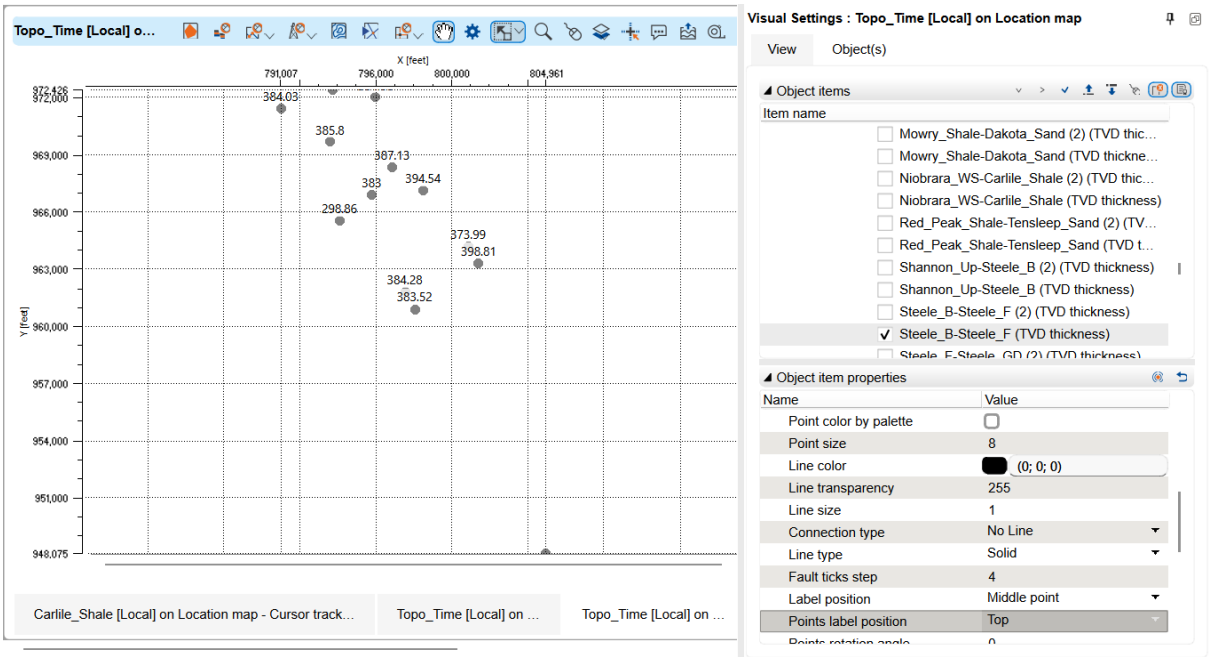
© 2026 Geomage 2003 LTD

When a time or depth 3D view[244] is active, selecting the **TVD thickness** check box shows the zone as an interval along each eligible well trajectory between its top and base markers, drawn in the colour of the zone itself; a well with either marker missing shows no interval, and a time 3D view also requires checkshot data for the well.

Displaying zones

Select the check box of a zone's **TVD thickness** item in the Data Manager to show the zone in the active view, and clear it to hide the zone. The matching check box on the Visual Settings[206] **Object(s)** tab carries the same state, so changing either check box changes the other.

On a Location map[235], the zone is a point at every well where both its top and base markers are picked. The point value is that well's TVD thickness, so enabling value labels places the thickness beside the well. The point is drawn in the colour of the zone itself, which **Change color** sets in the Data Manager. A well with either marker missing, or a well cleared in the marker dataset's **Wells** subsection, has no point for that zone.



On the **Object(s)** tab, open **Zones** for a Location map or **Zones 3D** for a 3D view. Each entry is named **<zone name> (TVD thickness)**. On a Location map, the usual object settings include point type, point size, colour and labels. The colour of the entry follows the zone: **Point color** under **Zones**, and **Color** under **Zones 3D**, hold the colour the zone carries in the Data Manager, while **Point color by palette** and **Color by palette** stay cleared, so the zone is drawn in that single colour instead of being coloured by its thickness values.

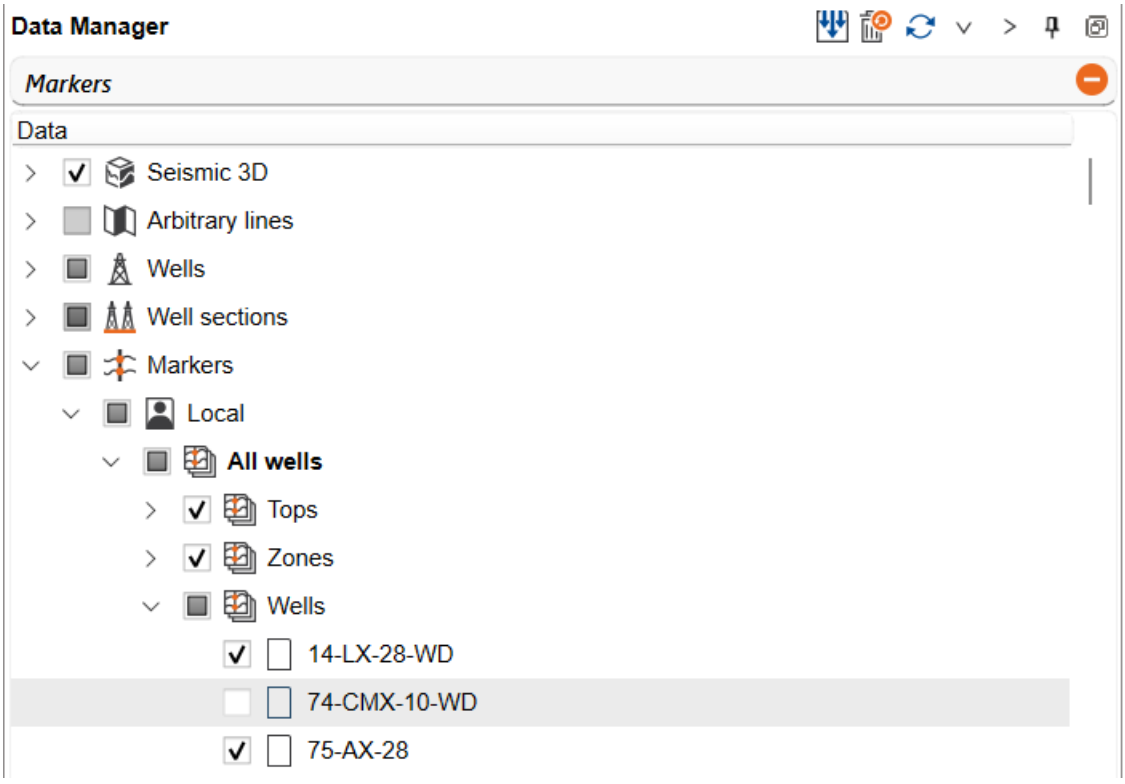
© 2026 Geomage 2003 LTD

Change color on a zone sets that colour once for every view. The zone is repainted immediately in every open view - the points on a Location map and the intervals along the well trajectories in a time or a depth 3D view - so the colour never has to be set again view by view. The colour belongs to the zone, so it is still there after the views and the project have been closed and opened again.

The link works in this direction only: a colour chosen on the **Object(s)** tab repaints the object in that view alone and leaves the colour of the zone in the Data Manager, and in every other view, unchanged.

Filtering marker displays by well

Every marker dataset has a **Wells** subsection that lists, by well name, every well containing markers in that dataset. Each well is selected by default. Clearing a well is a visualization filter: it hides that well's Tops and Zones from Location maps and 3D views and clears the corresponding Visual Settings entries. Selecting the well restores them. The marker and zone data are not deleted or changed, and the check state is saved with the project.



See Also

- Import markers 403
- Import markers structure 406

© 2026 Geomage 2003 LTD

3.2.1.7 Stratigraphy

Stratigraphy - contains the stratigraphic data
Right mouse button on the icon allows to:
- Load from XML
- Load default
- Clear
- Add folder

3.2.1.8 Horizons

Horizons - contains seismic picking
Right mouse button on the icon allows to:
- Add new horizon
- Clear horizons
- Sort by name
- Receive horizons
- Commit horizons

Right mouse button on a Time or Depth domain node additionally allows to:
- **Create thickness maps**
- **Create isopach maps** (Depth only)

These commands are enabled when the selected domain contains at least two horizons. **Create thickness maps** creates one map for each adjacent horizon pair. Time-domain results are stored under **Maps > Isochron maps** and named `Isochron_<lower horizon>-<upper horizon>`; depth-domain results are stored under **Maps > Isochore maps** and named `Isochore_<lower horizon>-<upper horizon>`. **Create isopach maps** stores depth-domain results under **Maps > Isopach maps**.
Right mouse button on an editable folder or subfolder under a Time or Depth node additionally allows to:
- **Add new horizon** — the new horizon is created directly in the selected folder, including a nested subfolder, instead of at the top of the Time or Depth node.

Right mouse button on a horizon additionally allows to:
- **Calculate Dip** — compute the dip (maximum slope) of the horizon surface.
- **Calculate Azimuth** — compute the dip azimuth (slope direction) of the horizon surface.

For a horizon the result is stored as a horizon-attribute map named `<horizon>_dip` or `<horizon>_azimuth`. The same **Calculate Dip** and **Calculate Azimuth** options are available on the right mouse button of a map, where the result is created as a separate map.

3.2.1.9 Faults

Faults - contains fault picking
Right mouse button on the icon allows to:
- Add new fault
- Clear faults
- Set meshing mode... (Sticks / Surface)
- Colorize

© 2026 Geomage 2003 LTD

- Export (To Charisma...)
- Receive faults
- Commit faults

Create a fault in a folder

To create a fault directly in an editable folder or subfolder, right-click the folder under **Faults** in the Data Manager 124 and choose **Add new fault**. Enter a fault name, select **Time** or **Depth**, and confirm. The new fault is placed in the selected folder.

Convert faults between time and depth

A selected velocity model 167 is required to convert faults between the time and depth domains.

- To convert all faults in a folder, right-click the **Time** or **Depth** domain folder, or a folder or subfolder containing faults from one domain, then choose **Convert to depth** or **Convert to time**.
- To convert several faults, select fault objects from the same domain, right-click the selection and choose the corresponding conversion command.

Conversion adds a copy of each included fault to the other domain. The original faults and **Free sticks** remain unchanged, and each converted copy keeps its source fault's color.
The command is disabled when no velocity model is selected and is not shown for a selection that mixes time and depth faults.

Rename a fault

To rename an editable fault, right-click it in the **Faults** branch of the Data Manager 124, choose **Rename**, enter a name that is not already used by another fault, and confirm. The command is disabled for faults that you cannot edit.

Assign Fault Type

Each fault can be classified by its geological type. Right-click an individual fault in the Data Manager 124 and choose **Assign Fault Type**, then one of:

- **Unknown** (default)
- **Normal**
- **Reverse**
- **Strike-slip**
- **Strike-slip + Normal offset**
- **Strike-slip + Reverse offset**

The selected type is saved as a property of the fault object. The Fault Modeling 181 wizard reads this type automatically for each fault and lets you edit it in the **Fault Type** column of its **Fault Inputs** tab.

© 2026 Geomage 2003 LTD

Fault type symbols on the Location map

On the Location map 235, a fault shown along the current horizon or along a depth map is drawn with the geological symbol of its assigned type. The symbols repeat along the fault line and use the fault line color. They stay on the same side of the line for the whole fault; that side is derived from the fault and horizon geometry.

- **Unknown** – a plain fault line, without symbols.
- **Normal** – short tick marks across the fault line.
- **Reverse** – filled triangular teeth.
- **Strike-slip** – pairs of half-arrowheads set on opposite sides of the fault line and pointing in opposite directions, showing the relative lateral movement of the two blocks.
- **Strike-slip + Normal offset** and **Strike-slip + Reverse offset** – the same paired half-arrowheads as **Strike-slip**.

The symbols are drawn on the Location map only, for the **Faults on current horizon**, **Faults on depth maps**, **Fault model faults on current horizon**, **Fault model faults on depth maps** and **Structural model faults on current horizon** items.
To change how often the symbols repeat, select the fault item in **Visual Settings** on the **Object(s)** tab and adjust **Fault ticks step**. The value is editable while **Line type** is set to **Fault**. Strike-slip arrows are drawn four times less often than ticks and teeth.

3.2.1.10 Topography

Topography - contains internal (computed) and external (loaded) topography surfaces.
Right mouse button on the icon allows to:

- Recreate internal topography
- Add folder

Right mouse button on an external topography item allows to:

- Clear external topography
- Load from file
- Save to file
- Download topography from server

Right mouse button on the internal topography item allows to:

- Recreate
- Update topography map
- Save to file

See Also

- Import topography 445

© 2026 Geomage 2003 LTD

3.2.1.11 Raster files

Raster files - contains project raster and image files (georeferenced backdrops, scanned maps and downloaded satellite imagery).
Right mouse button on the icon allows to:

- Add raster files
- Download satellite imagery
- Clear raster files
- Add folder

Right mouse button on a raster file additionally allows to:

- Delete raster

Add raster files imports one or more georeferenced image files into the project. **Download satellite imagery** opens the Satellite imagery downloader, which fetches a satellite backdrop and adds it here as a GeoTIFF.

See Also

- Satellite imagery downloader 161
- Import satellite imagery 449
- Image Data Formats 516
- Data manager 124

3.2.1.11.1 Satellite imagery downloader

The **Satellite imagery downloader** wizard fetches a satellite backdrop for the project area and stores it as a georeferenced GeoTIFF raster. The resulting raster can be used as a location-map background in 2D, map and 3D views, helping orient horizons, wells and polygons against real-world geography.

Launching the wizard

In Data manager 124 right-click the **Raster files** node and choose **Download satellite imagery**.

Input Parameters

- **Polygon** — select an existing project polygon to use as the download footprint, or pick **User defined** to type XY bounds manually.
- **Min X / Max X / Min Y / Max Y (m)** — bounding box of the request, used when the polygon is set to **User defined**.
- **Zoom level** — integer tile zoom. Default `15`; higher values give more detail but increase download size.

© 2026 Geomage 2003 LTD

- **Timeout (s)** — maximum time the downloader waits for the remote service before aborting. Default `60`.
- **CRS format** — coordinate reference system of the requested extent; used to reproject the result into the project CRS.

Running the download

The download runs in the background so the main application remains responsive. Progress and any server error are reported in the Output View. On completion the generated GeoTIFF is added to the Raster files folder in the Data manager and can be displayed as a backdrop in any view that supports raster imagery.

See Also

- Output View 219
- Data manager 124
- Location map 235

3.2.1.12 Shapefiles layers

Shapefile layers - contains loaded shapes and cultural data
Right mouse button on the icon allows to:
- Add shape files
- Load cultural points from text file
- Load custom points from text file
- Clear shape files
- Add folder

3.2.1.13 Maps

Maps - contains all grids loaded externally and generated by running velocity model.
Maps are separated by types:
- Depth maps
- Velocity maps
- Calculated maps
- Time maps
- Attribute maps
- Imported maps
- Spectral decomposition maps
- Property maps
- Isochore maps
- Isopach maps
- Isochron maps

© 2026 Geomage 2003 LTD

Right mouse button on the icon allows to:
- Clear maps
- Sort by name
- Add folder
- Receive maps
- Commit maps

Right mouse button on **type folder** allows to:
- Clear maps
- Load from text file
- Load from text file (irregular)
- Load from CPS-3 grid file
- Load from ZMap grid file
- Load from IRAP classic grid file
- Receive maps
- Commit maps

Right mouse button on an individual **Time** or **Depth** map allows converting it to the other domain:
- **Convert to depth** — available for a Time map; builds the corresponding depth map.
- **Convert to time** — available for a Depth map; builds the corresponding time map.

Converting a map between the time and depth domains requires a velocity model in the project. If no velocity model is available, g-Space shows the message *A velocity model is required to convert maps between time and depth. Please add a velocity model to the project first.* and the conversion is not performed.

Several maps can be selected together with **Ctrl + left click**. Right mouse button on the selection allows to apply one action to all selected maps at once:
- **Remove** — deletes all selected maps from the project in one action.
- **Save to text file**
- **Save to CPS-3 grid file**
- **Save to ZMap grid file**

The save options ask for one output folder and write a separate file for each selected map. The supported file formats and the export of a single map are described in Export maps 482.

3.2.1.14 Structural Models

The **Structural models** branch contains saved depth- and time-domain models, grouped by user and domain.

Right-click the branch, a user folder, or a **Depth**/**Time** folder and choose **Create new** to open the Structural Modeling 798 wizard. Starting from a domain folder presets that domain.

Right-click the **Structural models** branch and choose **Slicer** to drive the 3D slicing plane from the built models. **Show slicer** is a check item that turns the plane on and off and is always

© 2026 Geomage 2003 LTD

available; **Edit slicer parameters** and **Recalculate slicer parameters** are enabled only while the slicer is shown. When you switch the slicer on from this branch, or it was last driven from another branch, its extent is computed automatically from the layer volumes of every built structural model — the objects under each model's **Layers** folder — so at least one model must have been built with **Generate layers from horizon pairs** enabled; otherwise nothing is shown. **Recalculate slicer parameters** recomputes that extent from the same models and opens the slicer parameters wizard, while **Edit slicer parameters** opens that wizard for the current slicer without recomputing. There is a single shared slicing plane, so driving it here re-sources the same plane that the **Geo-bodies** and **3D Grids** branches use; see Geological modeling 758 for what the **Slicer** mode shows.

A built model organizes its results into these subfolders:

- **TIN Maps** — modeled horizon surfaces
- **Faults** — fault surfaces used by the model
- **Intrusions** — modeled intrusion surfaces
- **Layers** — volumes generated between horizon pairs
- **Fault Blocks** — separate blocks generated from a selected fault model

TIN Maps, **Intrusions** and **Layers** are always present under a built model and stay visible even when they contain nothing, so an empty **Layers** folder means that layer generation produced no volumes, not that the model itself is incomplete. **Faults** and **Fault Blocks** appear only when the model actually produced fault surfaces or fault blocks, so a model built without a fault model shows neither folder.

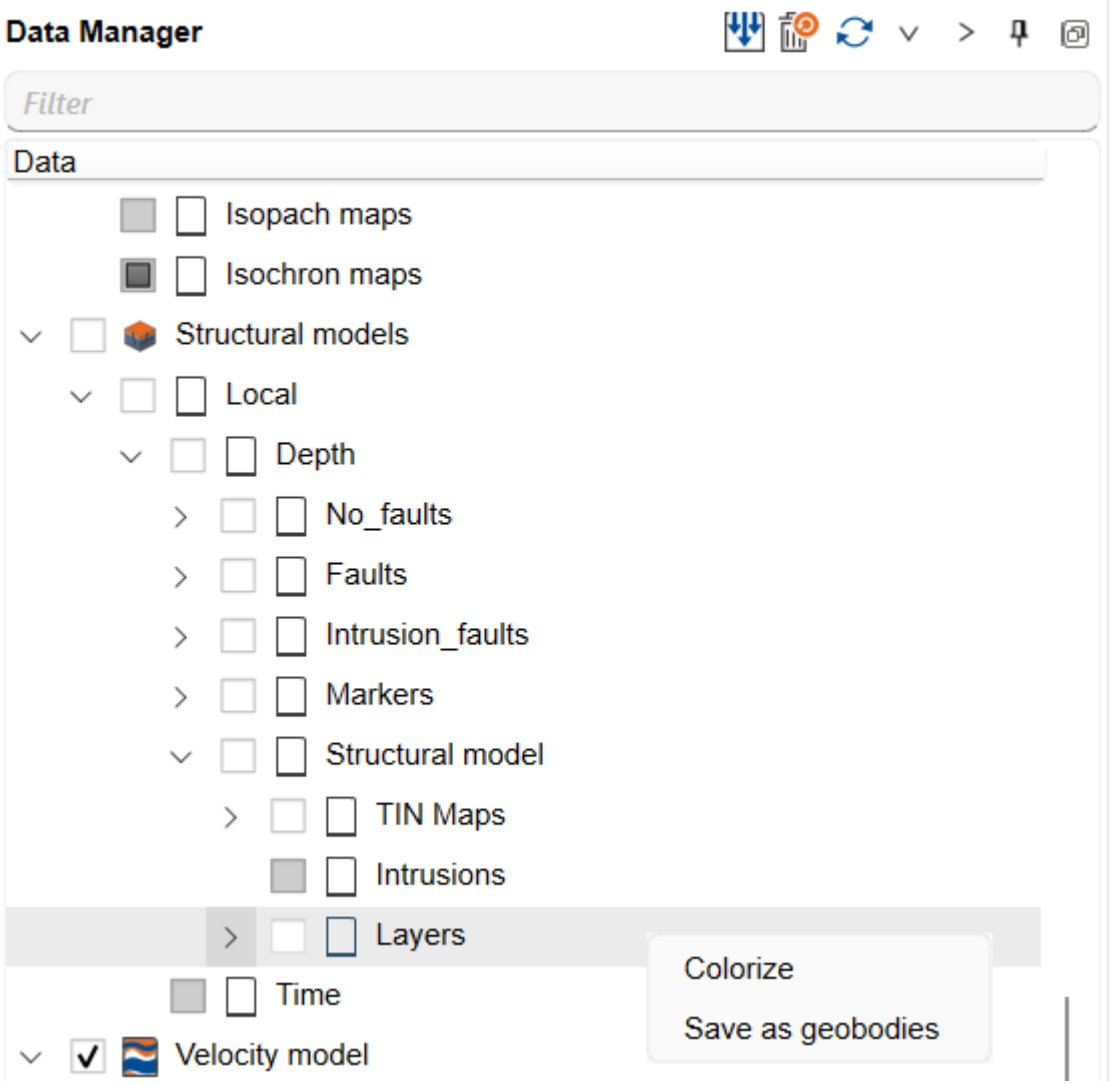
To give every object in a folder a different automatically generated color at once, right-click the model's **TIN Maps**, **Faults**, **Intrusions** or **Layers** folder and choose **Colorize**. The new colors are applied immediately, without a dialog, and repaint the Data Manager together with the views. On the **Faults** folder the command usually appears to change nothing, because each fault keeps the color it takes from the fault model rather than the one stored with the structural model. An individual fault under a model offers only **Extract fault data** and no **Set color**, so change a structural model's fault colors on the fault model, or on the interpreted fault of the same name, and not on the model; see Fault model 181. **Colorize** is not offered on the **Fault Blocks** folder; fault blocks are given distinct colors automatically when the model is built, and **Set color** on an individual fault block changes one of them.

To turn modeled fault surfaces into points, right-click the model's **Faults** folder and choose **Extract fault data** to include every fault in it, or right-click a single fault under that folder to include only that fault. The command opens the **Extract fault data** dialog with an **Output name** already filled in from the model name for the folder and from the fault name for a single fault; **Points** is the only **Output data type**, so the extraction always produces a point set under **Points** in the Data Manager. See Fault model 181 for the dialog options and for the same command on a fault model.

To reuse layer volumes as ordinary geobodies, right-click the model's **Layers** folder and choose **Save as geobodies** to copy every layer in that folder into the Geo-bodies 177 branch, or right-click a single layer and choose **Save as geobody** to copy just that one. Each copy takes the layer's geometry, color, age and class, and the structural model keeps its own layers, so the copy is an independent object — clearing the layer's check box does not hide the geobody,

© 2026 Geomage 2003 LTD

and clearing the geobody's check box does not hide the layer. From then on the copy behaves like any other geobody: it has its own check box in the Data Manager, an entry in the 3D view 244 and in the **Visual Settings** panel, and the standard geobody actions.



A copy is named after the layer as the Data Manager shows it. In that name a slash, a backslash, an opening square bracket, a space or a tab becomes an underscore and a closing square bracket is dropped, repeated underscores are collapsed, and a leading or trailing underscore is removed, so a layer shown as Object_1/Object_2 [Block 1] gives a geobody named Object_1_Object_2_Block_1. When a geobody of that name already exists, the **Geobody name collision** dialog offers **Overwrite existing objects**, **Create unique names** and **Cancel** (**Overwrite existing object**, **Create unique name** and **Cancel** for a single layer). A unique name is made by appending a space and a number, as in Object_1_Object_2_Block_1 1 and then Object_1_Object_2_Block_1 2. For the folder command the question is asked once, before anything is written, and the answer applies to every colliding name, so **Cancel** leaves the project unchanged.

A layer whose geometry is incomplete is skipped instead of converted: **Save as geobody** reports that the layer failed because its geometry is invalid and creates nothing, while **Save as**

© 2026 Geomage 2003 LTD

geobodies lists such layers under **Failed** in the summary it shows when it finishes. That summary also reports how many geobodies were saved, how many were overwritten and which ones were renamed to keep names unique. On an empty **Layers** folder, **Save as geobodies** reports that no layer objects are available to save as geobodies.
Each object under **TIN Maps**, **Faults**, **Intrusions**, **Layers** and **Fault Blocks** is listed in the Data Manager with its assigned color filling the object icon, so horizons, faults, intrusions, layers and fault blocks can be told apart by color directly in the tree. It is the object's own display color, the same one used in the 3D and section views, and a fault takes its color from the fault model. An object with no assigned color keeps the plain icon, and some interface themes show plain icons throughout.
Every object in these subfolders has its own check box. Select or clear a check box to show or hide just that object in the 3D view 244 and on section views; the check box on the **Structural models** branch hides the whole set. The same objects are listed under **Structural model 3D** on the **Object(s)** tab of the Visual Settings 206 panel, and that list stays in sync with the Data Manager check boxes.
The **Structural model 3D** item also has a **Display mode** property that limits what the model shows in the 3D view: **Full structure** (default) shows every object of the model, **Surfaces only** shows only the modeled horizon surfaces from **TIN Maps** and hides intrusions, layers and fault blocks, and **Faults only** shows only the fault surfaces. The mode is set on the parent **Structural model 3D** item and applies to all of its objects, so a cleared check box always wins: the mode only limits what the selected objects may show. The chosen mode is stored with the view. **Display mode** is a 3D view property and is not available for section views.
The same models are also displayed on inline, crossline and arbitrary line sections in both domains, where they are listed under **All structural model** on the **Object(s)** tab of the **Visual Settings** panel. Modeled horizon, intrusion and fault surfaces are drawn as their intersection lines with the section, while **Layers** and **Fault Blocks** are drawn as translucent filled bands that are split where the volume pinches out. Select a layer or fault block under **All structural model** to restyle its band: **Object item properties** then offers **Fill type** (**No fill**, **Solid**, a hatch pattern or a gradient), **Fill color**, **Fill color 2** for the second color of the gradient fill types, and **Transparency level**, which sets how translucent the band is; the band is redrawn immediately. By default a band is solid-filled in the object's own color and translucent. Each choice is kept per object and re-applied whenever the section is rebuilt, so a fill or transparency change is no longer lost when the view refreshes or you step to another line.
A model is displayed only in the views that match its own domain: a model stored in a **Depth** folder in the 3D view depth and on depth sections, and a model stored in a **Time** folder in the 3D view time and on time sections. No time-to-depth conversion is applied, so a time-domain model never appears in a depth view.
Double-click a model, or right-click it and choose **Open**, to restore its settings in the wizard. Use **Run** to rebuild or update its results and **Save** to store changed settings without running.
Other model actions include **Rename**, **Delete** and **Verify 3D model**.
Several models can be removed in one step. Select them in a **Depth** or **Time** folder, right-click the selection and choose **Remove**, the only command offered for a multiple selection. A single confirmation, *"Remove N selected structural models?"*, names the number of selected models;

© 2026 Geomage 2003 LTD

answer **No** to leave everything unchanged. Each model is then removed exactly as **Delete** removes a single one, so any Multi-Z surfaces published from it and any uncertainty realizations built from it disappear with it, and the Data Manager, the 3D view 244 and the **Visual Settings** object lists are refreshed at once. The command applies to the model entries only, not to their **TIN Maps**, **Faults**, **Intrusions**, **Layers** or **Fault Blocks** folders or to the objects inside them, and no dependency warning is shown here; that extra confirmation belongs to removing fault models (see Fault model 181).

See Also

- Structural modeling 798
- Geological modeling 758

3.2.1.15 Velocity model

Velocity model - contains depth / interval velocity models used for time-depth conversion.
Right mouse button on the icon allows to:
- Add velocity model
- Import velocity model
- Clear
- Add folder

Right mouse button on a velocity model additionally allows to:
- Set default
- Rename
- Remove
- Export to Seg-Y file

Import velocity model imports a velocity model from a SEG-Y file.

See Also

- Setting up the velocity model 660
- Import velocity model 678
- Export velocity model 679

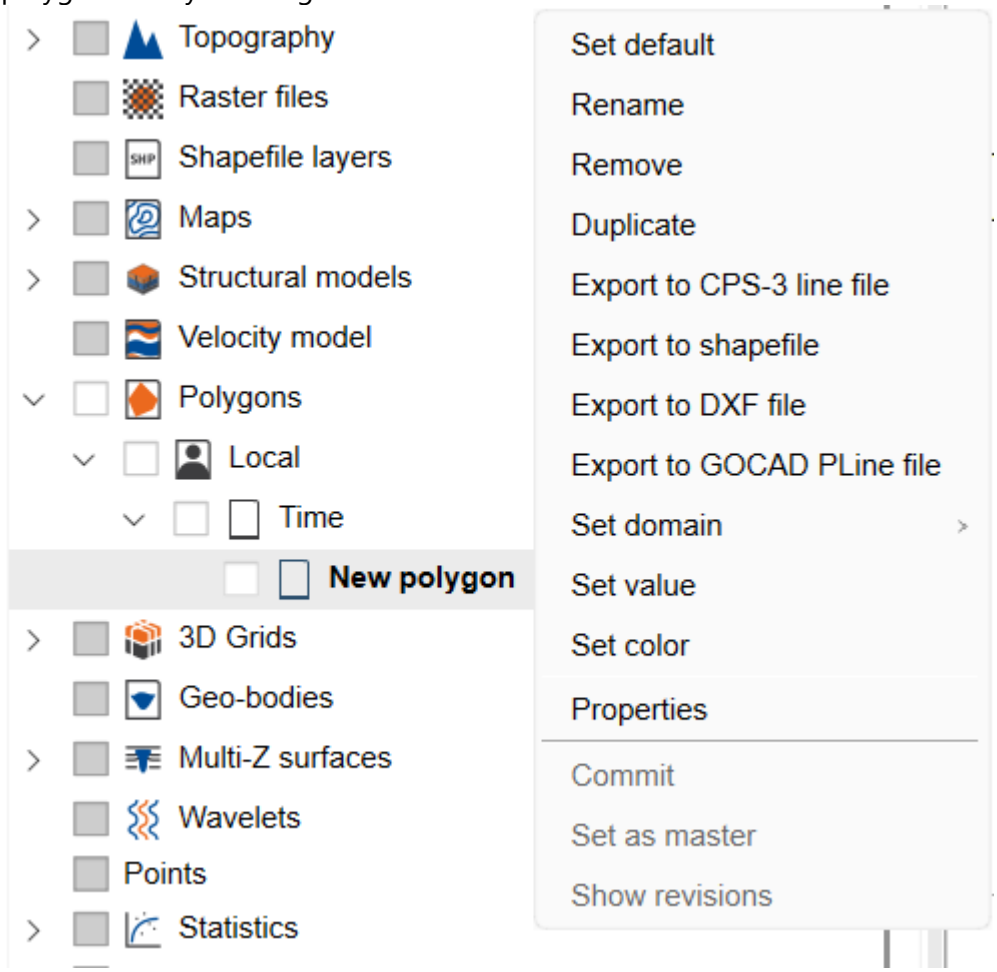
3.2.1.16 Polygons

Polygons - contains polygons created in g-Space
Right mouse button on the icon allows to:
- Add polygon
- Import > Import polygons from CPS-3 line file
- Import > Import polygons from DXF file
- Import > Import polygons from GOCAD PLine file
- Import > Import polygons from RESQML EPC
- Export > Export polygons to RESQML EPC
- Export > Export all polygons to CPS-3 line file

© 2026 Geomage 2003 LTD

- Export > Export all polygons to shapefile
- Export > Export all polygons to DXF file
- Export > Export all polygons to GOCAD PLine file
- Clear
- Receive polygons
- Commit polygons

A polygon can be stored either in the **Time** or in the **Depth** domain. Right mouse button on a polygon and choose **Set domain** to switch it between the **Time** and **Depth** domains.
Right mouse button on a polygon and choose **Set color** to give that polygon its own color. The standard color dialog opens; its **Reset** button clears the custom color and returns the polygon to its default appearance. The color is used in every view where the polygon is drawn, and on the Location map and on the section views it sets the polygon line and point color together. The polygon icon in the Data manager tree is filled with the same color, so the same polygon is easy to recognize both in the tree and in the views.



Polygons are displayed on the Location map 235, in the 3D view 244 and on the seismic section views: 2D line, inline, crossline and arbitrary line, in both the time and the depth domain.

© 2026 Geomage 2003 LTD

On a section a polygon is drawn as separate points at the places where its boundary crosses the section line, so a polygon that does not cross the current section is not shown. A polygon stored in one domain is converted with the current velocity model when it is displayed on a section of the other domain.
To show or hide the **Polygons** object in a section view, or to change its point type, color and size, use the Object 210 tab of the Visual Settings 206 panel.

3.2.1.17 3D Grids

3D Grids - contains 3D grids
Right mouse button on the icon allows to:

- Create new grid
- Import from ECLIPSE
- Import GOCAD SGrid
- Import GOCAD TSolid
- Import GOCAD Voxet
- Import from RESQML EPC
- Export all to RESQML EPC
- Clear
- Slicer (Show slicer, Edit slicer parameters, Recalculate slicer parameters)
- Add folder

Grid types

When you choose **Create new grid**, g-Space first asks for a **Grid type**:

- **S-Grid** — a structured, corner-point grid of hexahedral cells built from the standard grid parameters (interpolation and grid steps, angle, length, zones and layers). This is the default grid used across most workflows; see 3D grid creating 765.
- **Tetrahedral** — an unstructured tetrahedral mesh generated from an existing structural model, so it honours the model's structural and stratigraphic framework. You select the source structural model and a **Layering type**, then set only the parameter that applies to it: **Layers per zone** for **Proportional**, or **Layer thickness** for **Parallel to Top** or **Parallel to Bottom**; the other field is greyed out and its value is ignored. For the full procedure, see 3D grid creating 765. Tetrahedral grids are suited to complex volumes that structured grids cannot represent.

Grids can also be brought in from external formats. **Import from ECLIPSE** and **Import GOCAD SGrid** load structured grids, while **Import GOCAD TSolid** and **Import from RESQML EPC** load tetrahedral grids.

Time and depth grids

Every 3D grid has a **Time** or **Depth** domain, set from the data it is built from: grids built from time horizons or time maps are time-domain, while grids built from depth horizons or depth maps, or imported grids, are depth-domain. No time-to-depth conversion is applied, so a time grid keeps its native time geometry. Grids from older projects that predate this option load as **Depth**.

© 2026 Geomage 2003 LTD

In the **Data Manager**, the **3D Grids** branch groups grids under two folders, **Time** and **Depth**, and each grid appears under the folder that matches its domain. A time-domain grid is displayed in the 3D view in time, and a depth-domain grid in the 3D view in depth.

Showing grid zones in the 3D view

Each zone under a grid's **Zones** folder in the **Data Manager** has its own checkbox. Select or clear a zone's checkbox to show or hide that zone in the 3D view, independently of the other zones. This works for both S-grids and tetrahedral grids that have zones.

A tetrahedral grid built from a faulted structural model is meshed one fault block at a time, so its cells follow the model's top and bottom surfaces and stay inside the fault blocks instead of crossing the fault. Where a zone is split into more than one block, each block appears as a separate item under that zone in the **Data Manager**, named after the zone's top and bottom horizons followed by the block number, for example *<top horizon>/<bottom horizon> [Block 1]*. Each block item has its own checkbox, so a single fault block can be shown or hidden in the 3D view. A zone that is not split by a fault keeps a single entry, with no block items below it.

Showing grids on the Location map

Every tetrahedral grid in the project is also drawn on the **Location map**. The grid's top surface is projected to plan view, so the map shows the triangulated top-surface mesh of the grid, including all blocks of a faulted grid, rather than a plain rectangular outline. A grid that has not been built yet is not drawn.

Each footprint is part of the **Grid** object of the **Location map** and is linked to that grid's own node under **Data Manager** > **3D Grids**, so selecting or clearing the grid's checkbox shows or hides its footprint. The footprint takes the colour of the grid's first zone when that zone has one, and a neutral grey otherwise; like other Location map objects, its appearance is set in **Visual Settings**. The display also honours the **Show tet grids** option under **Performance** in Module properties 197, which is on by default.

For an S-grid, the **Location map** shows the lateral mesh of the current grid only.

Slicer

To cut the grids open in the 3D view, right-click **3D Grids** in the **Data Manager** and choose **Slicer** > **Show slicer**. This switches on a single slicing plane in the 3D view. The first time you show the slicer from **3D Grids**, g-Space fits the slicer box to the current 3D grid and opens the **Slicer parameters** wizard, where you can keep or change the box centre, start, steps, lengths and angle.

The plane cuts the grids that are currently shown in the 3D view along inline, crossline or depth. The part of the grid beyond the plane is clipped away and the exposed cut face is drawn as a real section of the grid: every cell is filled with its own value colour from the grid's current display settings, with no interpolation across a cell, and the cut face carries the same cell-boundary lines as the outer faces of the grid body. Drag the plane in the 3D view and the cut and its section face follow it immediately. The slicer works for both S-grids and tetrahedral grids.

© 2026 Geomage 2003 LTD

Show slicer is a check item and is always available. **Edit slicer parameters** and **Recalculate slicer parameters** are enabled only while the slicer is shown: **Recalculate slicer parameters** fits the slicer box to the current 3D grid again and then opens the wizard, while **Edit slicer parameters** opens the wizard for the existing plane without refitting it. If no 3D grid is loaded, g-Space reports *No 3D grids are loaded*.
There is a single slicer plane, shared with the **Geo-bodies** and **Structural models** branches, so showing it from **3D Grids** re-fits that same plane to the grids. For an introduction to the **Slicer** mode, see Geological modeling 758.

Extract maps from a grid zone

To create a 2D map from a built grid zone, in the **Data Manager** expand **3D Grids** > grid > **Zones**, right-click the zone and choose **Extract Map**. The **Extract map** wizard uses the selected grid and zone.
Under **Map type**, choose:

- **Top of zone** or **Bottom of zone** to extract a zone surface.
- For a depth grid, **Isochore map** or **Isopach map**; for a time grid, **Isochron map**.
- **Property maps** to create maps from properties that contain data in the selected zone. Select one or more **Property** entries and one or more **Statistic** values: Minimum, Maximum, RMS, Average, Median, or Most common for facies properties. A map is created for each compatible property-statistic pair.

The read-only **Storage location** shows the destination. Top and bottom surfaces are stored under Maps > Depth maps; the other results are stored under the corresponding Isochore maps, Isopach maps, Isochron maps or Property maps category.
The **Output name prefix** defaults to *<grid>_<zone>*, with spaces replaced by underscores. Result names append *_Top*, *_Bottom*, *_Isochore*, *_Isopach*, *_Isochron*, or the property and statistic names. Edit the prefix if required.
Click **Run** to create the maps, or **Save to workflow** to store the configured operation as a reusable workflow step. If the zone surfaces are unavailable, build the grid before running the extraction.

Extract Grid Property to LAS

This is the reverse of upscaling well logs into a grid. Right-click a single grid property (Data Manager > 3D Grids > grid > Properties > property) and choose **Extract Grid Property to LAS** to sample that property along well trajectories and store the result as a new LAS curve. It lets you compare the original log, the upscaled log and the modelled grid property along the same wells.
The **Extract Grid Property to LAS** wizard shows:

- **Source grid** and **Source property** — read-only, taken from the right-clicked property.
- **Wells selection** — **All wells** or **Specific wells** picked from the list.
- **Output suffix** — appended to the property name to form the curve name; it defaults to *_from_<Grid>*. For property Porosity on Grid1 the resulting curve is Porosity_from_Grid1.

© 2026 Geomage 2003 LTD

The extraction follows the grid geometry and cell values, so no separate sampling step is required. A LAS curve is created for each processed well and stored under that well, ready to display in the Well log and Well correlation views. Wells that cannot be sampled, for example wells that lie outside the grid, are skipped; after the run a single summary message lists each skipped well together with the reason. If a well already has a curve with the target name, that curve is overwritten without confirmation, so change the **Output suffix** if you want to keep the earlier curve. Extraction is available for S-grids only; tetrahedral grids are not supported.

3.2.1.17.1 3D Grid Quality Control

3D Grid Quality Control is a diagnostic tool that scans a 3D grid for cell geometry problems before the grid is used for property modelling, volumetrics or export. It reports the number of problem cells for each selected check, grouped by severity, so structural issues can be found and corrected early.

Purpose

The tool is designed to:

- Detect cells with invalid geometry (negative thickness, zero volume or negative volume)
- Flag cells that are too thin or collapsed for reliable modelling
- Identify cells with an extreme aspect ratio that can degrade interpolation and simulation
- Provide a per-check issue count with a **Critical** / **Warning** severity so problems can be prioritised

Requirements

Before running Quality Control:

- The grid must exist in the 3D Grids 169 section of the **Data Manager**.
- The grid must contain at least one **zone** with layers. A grid with no zones cannot be checked, and the tool reports *Grid has no zones to check*.
- Quality Control is available for standard 3D grids; it is not offered for tetrahedral grids.

Entry Point

Quality Control is launched from the Data Manager context menu:

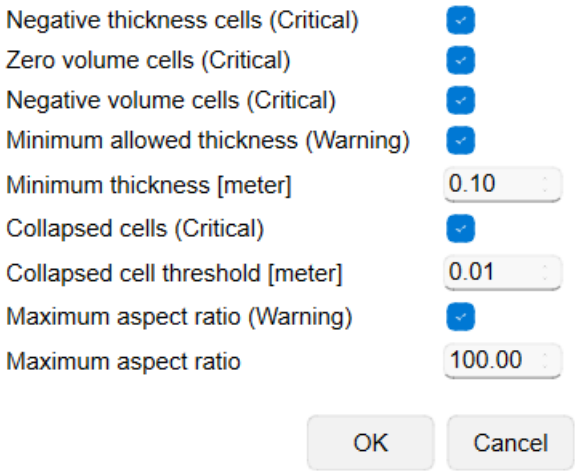
1. In the **Data Manager**, open the 3D Grids 169 section and locate the grid to check.
2. Right-click the grid and select **QC** from the context menu.

The **QC** window opens with a title that names the target grid, for example *QC for Grid: Grid 5*. The selected checks are evaluated across every zone and layer of the grid.

© 2026 Geomage 2003 LTD

The QC Window

The **QC** window lists the available checks as checkboxes. Checks that need a limit value show an input field beside them, and values are entered in the project's units. All checks are enabled by default.



Basic geometry checks

These checks need no parameters, and each flagged cell is reported as **Critical**:

- **Negative thickness cells** — cells whose bottom lies above their top (thickness < 0).
- **Zero volume cells** — cells with zero thickness, where the top and bottom coincide.
- **Negative volume cells** — cells with inverted, negative volume.

Threshold-based checks

These checks compare each cell against a limit that you can edit:

- **Minimum allowed thickness** — cells whose per-layer thickness is below the **Minimum thickness** value are flagged. Unit: project distance (m or ft). Default: **0.10**. Severity: Warning.
- **Collapsed cell threshold** — cells with an extremely small thickness, below the **Collapsed cell threshold** value, are treated as collapsed. Unit: project distance (m or ft). Default: **0.01**. Severity: Critical.
- **Maximum aspect ratio** — cells whose aspect ratio exceeds the **Maximum aspect ratio** value are flagged. The aspect ratio is the larger horizontal cell size divided by the cell thickness, max(dx, dy) / dz. Unit: dimensionless. Default: **100**. Severity: Warning.

© 2026 Geomage 2003 LTD

Running the check

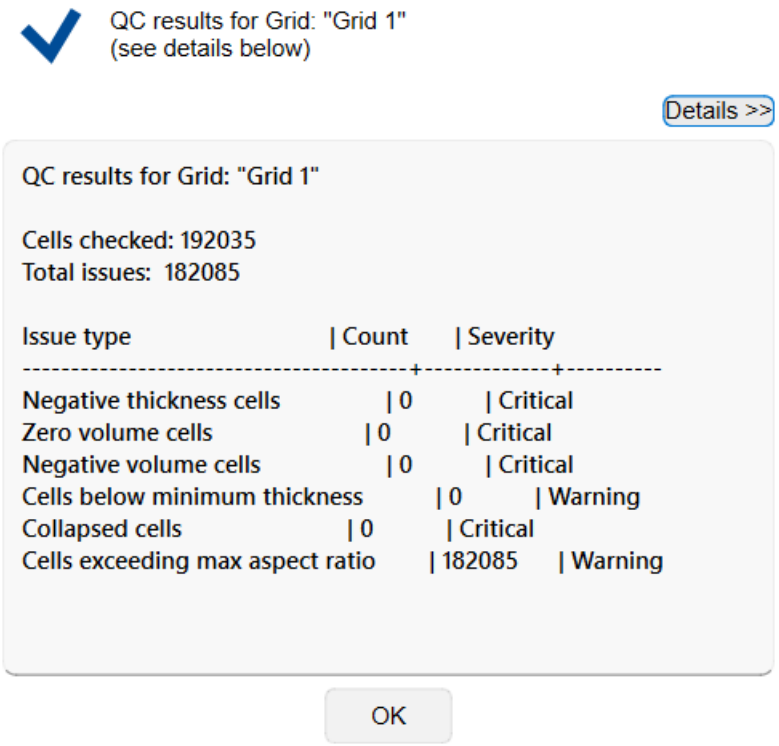
1. Enable or disable individual checks using their checkboxes.
2. For threshold-based checks, adjust the **Minimum thickness**, **Collapsed cell threshold** and **Maximum aspect ratio** values as needed.
3. Confirm the window to run Quality Control. The grid is scanned and the results report is displayed.

Reviewing Results

When the scan finishes, a results report is shown for the grid. It summarises:

- **Cells checked** — the total number of cells evaluated across all zones and layers.
- **Total issues** — the combined count of flagged cells from the enabled checks.
- A per-check table with the columns **Issue type**, **Count** and **Severity**. Only the checks you enabled appear in the table.

A high number of **Critical** issues (negative thickness, zero or negative volume, collapsed cells) indicates geometry that should be corrected before the grid is used for modelling. **Warning** issues (cells below minimum thickness, high aspect ratio) highlight cells that may reduce the quality of interpolation and simulation.



© 2026 Geomage 2003 LTD

Recommendations

- Run Quality Control after building or editing a grid's zones and layers, and again before property modelling or export.
- Keep all checks enabled for the first run to get a complete picture, then re-run with only the relevant checks to focus on specific problems.
- Treat **Critical** issues first: they usually point to inverted or degenerate cells that break volumetrics and simulation.
- Adjust the **Minimum thickness** and **Collapsed cell threshold** to match the vertical resolution of your model — very thin reservoir layers may legitimately fall below the defaults.
- Cross-check the counts against the grid's structure using **Show grid report**, which lists per-zone dimensions and volumes.

See Also

- 3D Grids 169
- 3D grid creating 765
- Properties modeling 776
- Object properties 221
- Workflows 535

3.2.1.17.2 Grid calculator

The **Grid calculator** creates a new property of a 3D grid from the properties that grid already contains. The expression is evaluated cell by cell, so it is a quick way to derive a property such as net-to-gross, a saturation or a permeability estimate from properties that were upscaled or modelled earlier. It works on both S-Grids and tetrahedral grids.

Opening the calculator

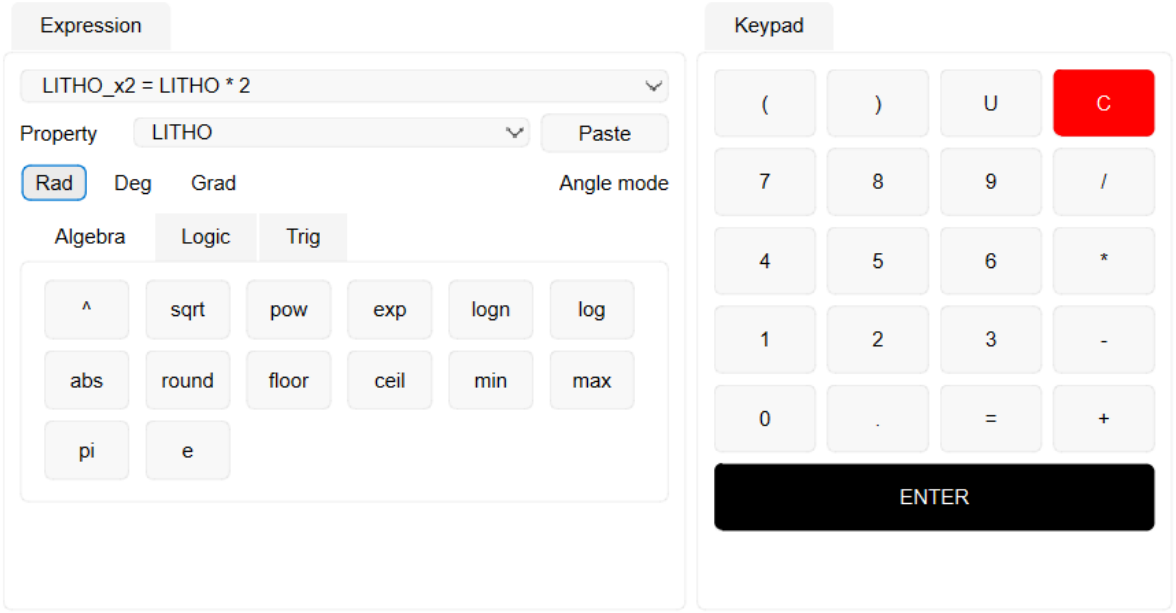
In the **Data Manager** expand **3D Grids**, right-click the grid you want to work on and choose **Open grid calculator** at the end of the grid context menu. The same tool is listed as **Grid calculator** in the Actions 214 panel; opened from there it works on the current grid.

The grid must already contain at least one property. If its **Properties** folder is empty, g-Space reports that the grid has no properties and that a property has to be added or upscaled first.

Writing the expression

© 2026 Geomage 2003 LTD

The name of the result and the formula are typed together in the **Expression** field, in the form **name = expression**. The part before the equals sign becomes the name of the new grid property, and the part after it is the formula, written with the names of the properties in that grid.



The main controls are:

- **Property** and **Paste** — select one of the grid properties and click Paste to insert its name into the expression at the cursor.
- **Algebra**, **Logic** and **Trig** tabs and the **Keypad** — insert mathematical functions, operators and digits. **U** inserts the undefined value and **C** clears the expression.
- **Angle mode** (**Rad**, **Deg**, **Grad**) — sets how trigonometric functions interpret angle values.

Refer to Trace headers math 521 for a description of the available mathematical functions.

The **Expression** field is a drop-down that remembers the last five expressions applied in this calculator, so a previous calculation can be reused without retyping it. The history is kept separately for each project, and the Grid calculator keeps its own list, so it does not share expressions with the other calculators.

Running the calculation

Click **ENTER** to run the calculation. If the name in front of the equals sign is missing, g-Space reports that the output name is not specified. If the grid already contains a property with that name, you are asked whether to replace it. Closing the window without clicking **ENTER** leaves the grid unchanged.

© 2026 Geomage 2003 LTD

The result is stored as a property of the same grid and appears under its **Properties** folder in the **Data Manager**, ready to display or to use as the input of a further calculation. A cell is left undefined where every property used in the expression is undefined, and where the formula does not produce a valid number.

3.2.1.18 Geo-bodies

Geo-bodies - contains created geobodies
Right mouse button on the icon allows to:

- Load from file
- Class catalog...
- Group by class
- Clear
- Colorize
- Slicer (Show slicer, Edit slicer parameters, Recalculate slicer parameters)
- Add folder

Creating a 3D geological model

Geo-bodies are built with the **Create 3D geological model** wizard, launched from the **Geo-body** group of the **Reservoir** ribbon tab. On its **General** tab you define the model layers (Name, Top, Bottom, Age) and the bounding polygon, and on the **Faults** tab you choose which faults cut the model.

Fault models

On the **Faults** tab, in addition to selecting individual faults you can enable whole fault models under **Fault models**. Only depth-domain fault models are listed. When a fault model is checked, its fitted fault surfaces are used to cut the geological model together with any individually selected faults.

Displaying a geo-body in the 3D view

Click a geo-body in the 3D view 244 to select it, then click the settings (gear) button in the top-right corner of the view to open the **Geomage 3D Settings** window. Its **Inspector** tab shows the properties of the selected object, including a **Display Mode** list that sets how the body is drawn:

- **Solid** - the default: the body is drawn as filled surfaces.
- **Transparent** - the body is drawn filled but rendered so that objects behind it stay visible when **Opacity** is set below 1.
- **Outline** - only the edges of the body are drawn.
- **Points/Voxels** - only the nodes of the body are drawn.
- **Surface Mesh** - the edges and the nodes are drawn together.

© 2026 Geomage 2003 LTD

Extracting points from a geo-body

To turn the geometry of a single geo-body into point data, right-click that geo-body in the Data manager 124 and choose **Extract points**. The nodes of its surfaces are collected, repeated coordinates are dropped, and the result is stored as a new object under Points 185, where it behaves like any other point set.
The new point set takes the name of the source geo-body with the suffix **_points** - a geo-body named **Geobody_1** gives **Geobody_1_points**. If that name is already in use, a number is appended (**Geobody_1_points 1**, **Geobody_1_points 2**, and so on), so an earlier extraction is never overwritten.
Besides the **X** and **Y** coordinates, the set carries a **Depth** attribute holding the Z value of each node. That attribute is already assigned the depth role, so the extracted points appear at the correct depth on sections and in 3D without further setup.

3.2.1.19 Multi-Z Surfaces

Multi-Z surfaces - contains multi-valued (TIN / faulted) surfaces that can carry several Z values per XY location.
Surfaces are organised under **Time** and **Depth** domain nodes and are handled in the same way as faults: each surface has a check box in the Data manager that controls its visibility and expands into separate nodes for the triangulated mesh and for the picked sticks, one surface at a time is the **active** surface used for picking and editing, and colour and other visual settings match those used for faults.
The entry for a surface in the Data manager is drawn in the colour assigned to that surface with **Set color**, or with **Colorize** when a group of surfaces is given distinct colours, so the same surface is easy to recognise in the location map, the 3D view and on seismic sections. The entry follows the colour as soon as it is changed, and a surface with no colour assigned keeps its default appearance. Only the surface entry itself carries the colour, not its **Surface** and **Sticks** nodes.
Right mouse button on the icon allows to:

- Create new
- Export all to RESQML EPC
- Clear
- Colorize
- Sort by name
- Add folder

Right mouse button on a surface additionally allows to:

- Export to GOCAD TSurf
- Set color
- Rename
- Remove
- Convert to depth / time
- Convert to regular map

© 2026 Geomage 2003 LTD

- Recreate surface
- Clear sticks
- Mesh optimization

Import into a Multi-Z surface (**Import from GOCAD TSurf**, **Import from RESQML EPC**) is offered on the Time / Depth domain sub-node.

Displaying and editing surfaces

- Tick a surface's check box in the Data manager to display it. A checked surface is projected onto inline, crossline and arbitrary-line seismic sections (in both time and depth), and drawn in the location map view and in the 3D view of its own domain: a surface under **Time** appears only in a time 3D view, a surface under **Depth** only in a depth 3D view. Picked sticks are drawn in 3D under the same rule. The check box on the surface shows or hides it as a whole, while the check boxes on its **Surface** and **Sticks** nodes control the triangulated mesh and the individual sticks separately.
- Select a surface to make it the **active** surface. The active surface is the target for interactive stick picking on seismic sections, exactly as for faults.
- Build the surface by adding sticks on a seismic section, use **Clear sticks** to remove them and **Recreate surface** to rebuild the triangulated mesh from the current sticks. Sticks are joined by their position on the surface, not by the order in which they were picked, so sections can be picked in any order and an earlier inline or crossline can be revisited without creating a long connection across the surface; a stick picked in the opposite direction is reversed automatically so that the mesh does not twist.

Sticks under a Multi-Z surface

Every Multi-Z surface can be expanded in the Data manager: it contains a **Surface** node for the triangulated mesh, and a surface that was built by picking also has a **Sticks** node that lists every picked stick by name (**Stick 1**, **Stick 2**, and so on). The sticks are listed one after another and are not grouped by inline, crossline or slice; the current stick of the active surface is shown in bold.

Each of these nodes has its own check box, so the mesh and every individual stick can be shown or hidden separately. Sticks are also a separate entry in Visual Settings 206: **All Multi-Z sticks** on seismic sections and the location map, and **Multi-Z sticks 3D** next to **Multi-Z surfaces 3D time** / **Multi-Z surfaces 3D** in the 3D views.

- Select a stick to make its surface the **active** surface and that stick the current one for picking.
- Right mouse button on a stick allows to **Rename** or **Remove** it. Removing a stick rebuilds the surface from the remaining sticks; the surface itself is kept.
- Drag sticks onto another Multi-Z surface in the Data manager to move them - a single stick, a multiple selection, or the whole **Sticks** node. The sticks keep their names, and both the

© 2026 Geomage 2003 LTD

source and the destination surface are rebuilt. A move is accepted only between surfaces of the same domain: time to time and depth to depth.

Convert surfaces between time and depth

A selected velocity model[167] is required to convert a Multi-Z surface between the time and depth domains.
Right-click a surface in the Data manager[124] and choose **Convert to depth** for a surface under **Time**, or **Convert to time** for a surface under **Depth**. The command is disabled when no velocity model is selected.
Conversion adds a copy of the surface under the opposite domain node, keeping the source surface's name and colour; if that name is already taken, a numeric suffix is added. The original surface remains unchanged.
For a surface built from picked sticks, the sticks are converted and the mesh is rebuilt from them, so the copy stays editable in the new domain. Parts of the surface that fall outside the velocity model coverage are left out of the copy, and a message is shown if no part of the surface can be converted.

See Also

- GOCAD Data Formats[519]
- Model Data Formats[522]

3.2.1.20 Wavelets

Wavelets - contains wavelets extracted or estimated by processing procedures (used for synthetics, well ties and inversion).
Right mouse button on the icon allows to:
- Clear
- Sort by name
- Add folder

Right mouse button on a wavelet additionally allows to:
- Open
- Rename
- Remove

Wavelets are produced by processing procedures rather than created from this node, so the icon menu has no **Add** action.

See Also

- Well-Tie[567]

© 2026 Geomage 2003 LTD

3.2.1.21 Fault model

Fault model - contains 3D fault-network models built from fault picking (used by structural modeling).
Right mouse button on the icon allows to:
- Create new
- Add folder

Right mouse button on a fault model additionally allows to:
- Open
- Rename
- Convert to Multi-Z surfaces
- Colorize
- Delete
- Extract fault data

Several fault models under **Fault model** → **Local** → **Depth** or **Time** can also be selected together. Right mouse button on such a selection offers a single command, **Remove**, which deletes all of the selected models: g-Space first asks you to confirm the removal of the selected models and, when structural models were built on any of them, reports how many structural models reference the selection and asks whether to remove it anyway. Answering **No** to either question leaves the project unchanged. Confirming deletes each selected model together with the Multi-Z surfaces published from it, and unlinks every structural model that used a removed fault model: such a structural model keeps its existing results, and when it is reopened the **Structural Modeling** wizard shows **Ignore faults** as its **Fault model**. The commands above remain single-selection, and **Remove** is offered on the fault models themselves, not on the individual modeled fault surfaces.
Right mouse button on an individual modeled fault surface allows to:
- Set color
- Extract fault data

Create new opens the **Fault Modeling** wizard.

Extract fault data

Choose **Extract fault data** on a fault model to create one point set from all of its modeled fault surfaces, or on an individual modeled fault surface to include only that surface.
In the **Extract fault data** dialog, enter an **Output name** and adjust **Add source fault name as attribute**, **Remove duplicate points** and **Skip faults with empty geometry (report in results)** as needed. Both **Run** and **Save** create the point set under Data Manager 124 → **Points**. **Save to workflow** adds the configured extraction to the project workflow 535 without creating the point set immediately.

Fault Modeling Wizard

© 2026 Geomage 2003 LTD

The **Fault Modeling** wizard builds modeled fault surfaces from interpreted fault data stored in the Data Manager. At the top of the dialog choose the **Domain** (**Depth** or **Time**) and enter a **Name** for the model. The workflow is organized into three tabs.

Fault Inputs

Select which faults take part in the model and how their data is used.

- **Available Inputs** (left panel) lists the faults from the Data Manager. The **Faults** and **Multi-Z surfaces** checkboxes filter what is shown.
- Use **Add** → and ← **Remove** to move items between the available list and the **Selected Faults** table.
- The **Selected Faults** table has columns **Fault Name**, **Points**, **Sticks**, **Maps**, **Markers** and **Fault Type**.

Each data-type cell (Points, Sticks, Maps, Markers) sets how that input is used. On an interpreted fault, only the data types actually present are editable:

- **No data** (grey) - the data type is not used.
- **Soft** (green) - guiding constraint; the surface is attracted to this data but may deviate from it for a smoother result.
- **Hard** (blue) - strict constraint; for **Points**, **Maps** and **Markers** the surface is forced to pass exactly through this data, and for **Sticks** it follows the picked sticks as closely as possible.

On a fault modeled from picked sticks, the **Sticks** cell also decides how the fault surface is built. With **Sticks** set to **Soft**, the surface follows the general fault trend fitted through the picked data instead of reproducing every stick, so noisy or irregular individual picks no longer pull it out of shape. With **Sticks** set to **Hard**, the surface keeps following the picked sticks as closely as possible, even when that leaves it rougher. The **Points**, **Maps** and **Markers** cells keep the constraint meaning above and do not switch this behavior.

Each **Hard** point of an interpreted fault's **Points**, **Maps** and **Markers** data becomes a fixed node of the modeled surface, and **Smoothing (%)** cannot move it afterwards. Set **Markers** to **Hard** when the modeled fault surface must intersect the fault markers picked in wells. Hard points that lie closer together than the smaller of **Step X** and **Step Y** are reduced to one fixed node per grid cell.

A **Multi-Z surface** row is treated as one already-interpreted geometry rather than picks, maps and markers: its **Points** cell is always editable and carries the single **Soft** or **Hard** mode of the whole surface, starting at **Soft**, while **Sticks**, **Maps** and **Markers** show a dash (—) and do not apply; the **Fault Type** cell stays editable as for any other input. With **Hard**, the geometry is used as it stands: the outer edge of the surface is held in place and **Smoothing (%)** reshapes only the interior, so a Multi-Z surface lofted from just two picked sticks is left essentially unsmoothed. With **Soft**, the geometry is a flexible constraint and **Smoothing (%)** may move the edge in

© 2026 Geomage 2003 LTD

depth or time as well, while the surface keeps its map-view outline. **Extend H** and **Extend V** apply in both modes. Only these two modes are meaningful here: leaving the cell at **No data** does not drop the surface, which is still built with its edge held in place exactly as for **Hard**.

The **Fault Type** column (**Normal**, **Reverse**, **Strike-slip**, **Unknown**, **Strike-slip + Normal offset** or **Strike-slip + Reverse offset**) is read from each fault's stored type and can be edited here. See Assign Fault Type 158 for setting the type directly on the fault object.

Surfaces

Define surface-generation parameters, one row per selected fault. Columns: **Fault Name**, **Step X**, **Step Y**, **Smoothing (%)**, **Shape**, **Extend H** and **Extend V**.

- **Step X** / **Step Y** - grid sampling in project units (meters or feet).
- **Smoothing (%)** - how strongly the generated surface is smoothed (0-100). This is a separate setting from the **Soft** / **Hard** input mode, and works differently with each.
- **Shape** - **Ellipse**, **Rounded boundary** or **Input shape**. Input geometry is always preserved.
- **Extend H** / **Extend V** - how far the surface extends beyond the input data, in project units. Disabled when **Input shape** is selected.

On a fault modeled from picked sticks, **Smoothing (%)** acts on the result of the input mode chosen on the **Fault Inputs** tab. With **Hard** input it only reduces roughness, and the data the surface must pass through stays in place; with **Soft** input it also flattens the fitted trend, from gently curved at low values towards almost planar at 100%, so **Soft** input with a high value gives a very smooth, nearly planar surface that still preserves the main fault trend.

Fault Intersections

Intersections between selected faults are detected automatically. For each pair the table shows **Fault**, **Intersects With** and **Dominant Fault**. The **Dominant Fault** drop-down in each row offers **Keep both** first, followed by the names of the two inputs in that row; a **Multi-Z surface** used as an input is listed there like a fault and can be selected as the dominant object. Every newly detected pair starts with **Keep both**, which applies no cut at that intersection and leaves both surfaces full. Select one of the two names for a pair to make that input dominant: it continues through the intersection and the other one is truncated at it, so a **Multi-Z surface** selected as dominant is preserved and the fault crossing it is cut back to that surface.

Running and Saving

Select the **Fault Inputs**, **Surfaces** and **Fault Intersections** tabs directly. **Run** generates the modeled fault surfaces using the selected data constraints, surface parameters and intersection rules, stores the model under Data Manager 124 → **Fault model** → **Local** → **Depth** or **Time** (matching the wizard's **Domain**) → the model **Name**, and keeps the wizard open. It is enabled

© 2026 Geomage 2003 LTD

after you provide a unique model name and select a fault with at least one input type set to Soft or Hard, or select a Multi-Z surface. **Save** stores the current model settings in the same branch without recalculating and also keeps the wizard open. The open wizard stays attached to the model it has just committed: adjust the fault inputs, surface parameters or intersection choices and click **Run** again to recalculate and update that same model in place, keeping its Data Manager entry and applying the current **Name** text to it, instead of creating a second model. A calculated model is visible in the 3D View 244 and can be reopened with **Open**, which restores its fault inputs - including the **Soft** or **Hard** mode of a **Multi-Z surface** row, in its **Points** cell - and **Surfaces** values together with the **Dominant Fault** choice saved for every detected pair - one of the two names, or **Keep both** - so a saved model can be reviewed, edited and re-run without repeating those intersection decisions. A model saved by an older product version that used a **Multi-Z surface** input is the one exception: every pair involving that surface reopens as **Keep both**, so make the dominant choice again before running.

Each selected input that produces geometry becomes a child node under the model, named after that input. A **Multi-Z surface** input is built with the **Step X**, **Step Y**, **Smoothing (%)**, **Shape**, **Extend H** and **Extend V** values of its **Surfaces** row and takes part in **Fault Intersections**; its child node carries the Multi-Z surface's own name, and the surface used as input is left unchanged under **Multi-Z surfaces**. These child nodes are the individual modeled fault surfaces on which **Set color** and **Extract fault data** operate.

Each modeled fault surface is listed in the Data Manager with its colour filling the object icon, and the same colour is used for that surface in the 3D View 244 and on section views. It is the colour given to the modeled fault surface with **Set color**, or assigned by **Colorize** on the fault model; when none has been set there, the modeled fault surface takes the colour of the interpreted fault of the same name under **Faults**, so a fault keeps the colour it was given when it was picked. A colour set explicitly for that surface in a view still takes precedence. See Structural models 163 for how a structural model built from this fault model uses these colours.

Save to workflow adds the current settings as a Fault Modeling task in the project workflow 535 without running the model. Use **Close** to exit the wizard.

Running or saving a fault model does not automatically publish its per-fault surfaces under **Multi-Z surfaces**. To publish them, right-click a calculated model in the Data Manager and choose **Convert to Multi-Z surfaces**. After this explicit conversion, each later run of the same model refreshes its Multi-Z surfaces. Multi-Z surfaces already stored by older product versions are preserved.

See Also

- Structural modeling 798
- Fault picking 627

© 2026 Geomage 2003 LTD

3.2.1.22 Points

Points is a Data manager 124 object type for storing scattered point data - for example scatter points, culture points, measured values or computed results. Each Points object (point set) stores an X and Y coordinate for every point plus any number of user-defined attributes such as Z, time, porosity or velocity.

Purpose

A Points object is a table of points. Every point always has the two required plan coordinates and can carry an unlimited number of additional attributes:

- **X** and **Y** - the required plan coordinates of each point.
- **Attributes** - any number of extra columns (for example Z, time, porosity, velocity), each with its own name and data type.

Each attribute has one of three data types, chosen in the **Add attribute** dialog:

- **Int** - whole-number (integer) values.
- **Float** - floating-point (double) values.
- **String** - free text (up to 50 characters).

Creating and loading Points

A Points object is created by importing an ASCII text file. In the Data manager 124, right-click the **Points** branch (or a user folder inside it) and choose **Import from ASCII...**, then select the text file.

The **Import points** column-mapping wizard opens. Set the separator, header row and start row, map the first two columns to **X** and **Y** (both required), map the remaining columns to attribute fields and pick a type for each, and enter a **Set name**.

Supported input formats are comma-separated (`.csv`), tab-separated (`.tab`) and plain text (`.txt`) files. The file has a header row of field names followed by one row per point; the first two columns are X and Y and the rest become attributes:

- `X Y Z Porosity`
- `512340.0 6017820.0 1875.4 0.18`

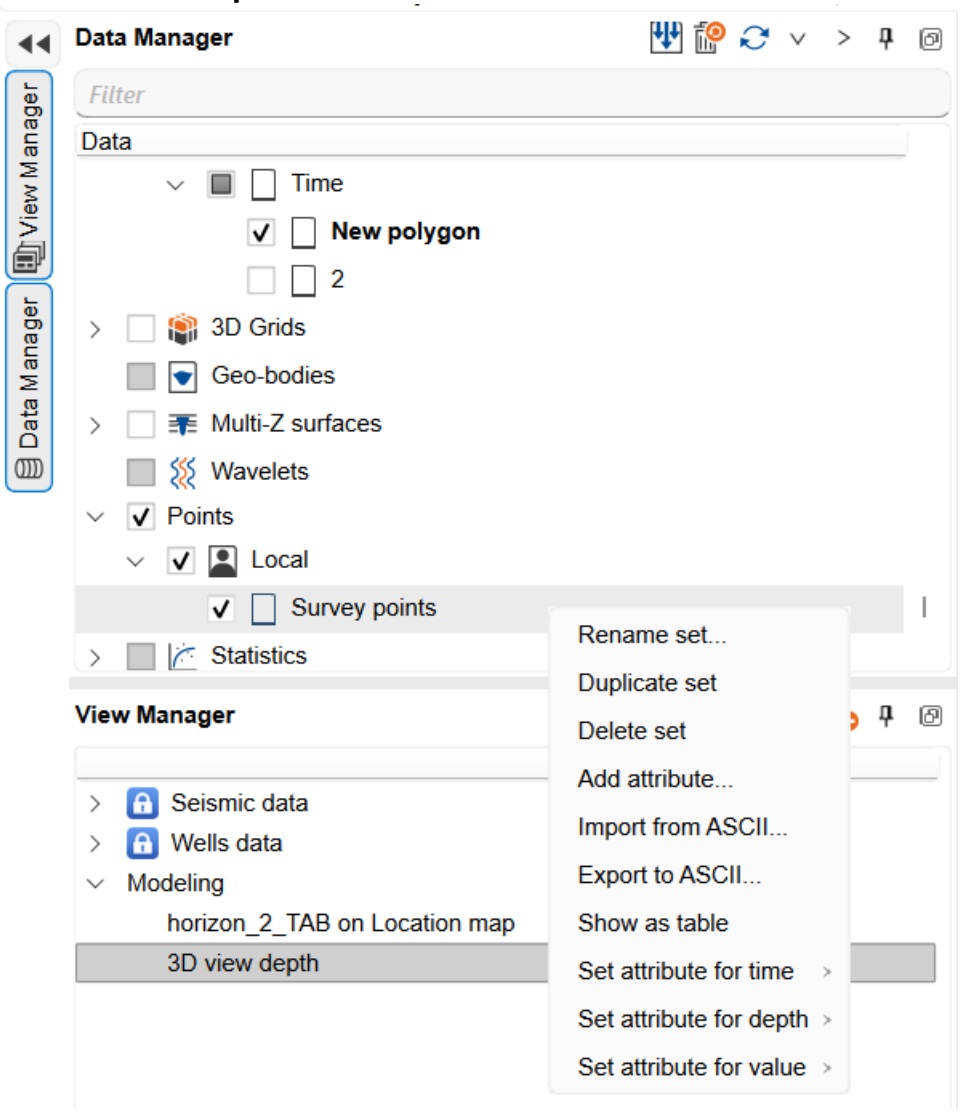
Point sets can also be exchanged in RESQML through the **Import from RESQML EPC...** and **Export all to RESQML EPC...** commands. See also Import points 460.

© 2026 Geomage 2003 LTD

A point set can also be created from existing model geometry. Right-click a geo-body in the Data manager 124 and choose **Extract points**: the geo-body geometry is written to a new point set under **Points**, named after the source geo-body. See Geo-bodies 177.

Attributes

Right-click a point set to manage its schema. **Add attribute...** opens a dialog with a **Name** field and a **Type** list (**Int**, **Float**, **String**; default **Float**). The same set context menu also offers **Rename set...**, **Duplicate set**, **Delete set** and **Show as table**.



Any point set can also tag individual attributes with a display role through the context-menu submenus:

© 2026 Geomage 2003 LTD

- **Set attribute for time** - the chosen attribute supplies the time (Z) value, so the points are positioned in time on 3D views and seismic sections.
- **Set attribute for depth** - the chosen attribute supplies the depth (Z) value for depth-domain display.
- **Set attribute for value** - the chosen attribute drives the colour (palette) of the points.

Only numeric attributes (**Int** or **Float**) can take these roles; **String** columns are not offered. Choosing the attribute that already holds a role clears that role.

Viewing and editing

A point set is shown in the Table view 285 as a grid whose first two columns are **X** and **Y**, followed by one column per attribute. Editing a cell updates the point immediately, and coordinate edits refresh the map, section and 3D displays.

Right-click a point set and choose **Show as table** to load it into the **Properties table** item of the Table view 285; the caption of that item then reads **Properties table: Points -** followed by the set name. Its columns are a read-only **Point index** column numbering the points from 1, then **X** and **Y**, then one column per attribute in the order the attributes are defined. An attribute that carries units shows them in its header in parentheses, for example **Depth (m)**.

Unlike the grid described above, this table does not pass an edit on at once: a value typed into an **X**, **Y** or attribute cell is only held in the table until **Save edits** in the header of the Table view writes it to the point set, and **Discard edits** drops it again. See Table view 285 for the full editing rules, including how a refused value is reported.

Points are also drawn on maps and seismic sections. When an attribute is assigned the time or depth role the points appear at the correct Z on sections and in 3D; when an attribute is assigned the value role the points are colour-coded by it.

Using Points

Point attributes behave like Marker attributes, so they can be reused elsewhere in g-Space:

- **Calculator** - attributes can be referenced in the calculator the same way as marker 551 attributes, to derive new values or filter points.
- **Cross-plots** - any two attributes can be plotted against each other in a cross-plot 745 to inspect trends and correlations.

© 2026 Geomage 2003 LTD

Exporting Points

Right-click a point set and choose **Export to ASCII...** to write it to a delimited text file. The header row lists **X**, **Y** and every attribute name, and each following row is one point. Choose **Tab separated values (*.tab)** or **Comma separated values (*.csv)**; full double precision is preserved so import - export - import round-trips are lossless.
The whole Points branch can also be written with **Export all to RESQML EPC...**. See also Export points 499.

Recommendations

- Keep X and Y in a consistent coordinate reference system and units across all point sets.
- Choose the smallest adequate type: **Int** for counts or codes, **Float** for measurements, **String** for labels.
- Assign a **Set attribute for time** or **Set attribute for depth** attribute before displaying points on sections so they land at the correct Z.
- Assign a **Set attribute for value** attribute to colour-code points by the property of interest.
- Use **Duplicate set** before bulk edits to keep an untouched copy of the original data.

See Also

- Polygons 167
- Import points 460
- Export points 499
- Data manager 124
- Table view 285
- Cross-plot 745
- Working with Markers 551

3.2.2 Filter wizard

The **Filter wizard** creates reusable filters that restrict well-related operations (property calculators, well-based gridding, petrophysics selections, well export) to a subset of the project's wells. In the current build, filters only support Wells data. Two modes are available:

© 2026 Geomage 2003 LTD

- **By expression** (automatic) — the filter is defined by a regular expression (ECMAScript syntax, case-insensitive) matched against the well name and, if present, its LAS curve names, checkshot names, comments, and raster/inclinometry names; membership is re-evaluated automatically whenever the well list changes.
- **Manual** — the user picks items explicitly from a tree of all project objects.

Launching the wizard

In Data manager 124, expand **Wells** and right-click the **Filters** folder, then choose **Add filter** to create a new filter, or **Edit filters** to open the wizard on the existing set.

Input Parameters

- **Filter name** — unique label. The wizard refuses duplicates.
- The filter name is edited in place in the **All filters** list; the editor turns red with a tooltip while the name is empty or already used.
- **Mode** — **By expression** or **Manual**.
- **Expression editor / tree view** — both sit together in the left-hand **Filter parameters** group; the expression text also narrows which rows the tree displays. In **By expression** mode the tree's checkboxes are hidden (membership comes from the regular expression); in **Manual** mode checkboxes appear so items can be picked explicitly.
- **All filters** list — right-click an existing entry for **Copy** or **Remove**, or right-click empty space for **Create new filter**, so multiple filters can be maintained in parallel.

Results

Saved filters appear under Filters in the Data manager and can be applied to any procedure that accepts a filter argument (exports, property modelling, QC tools).

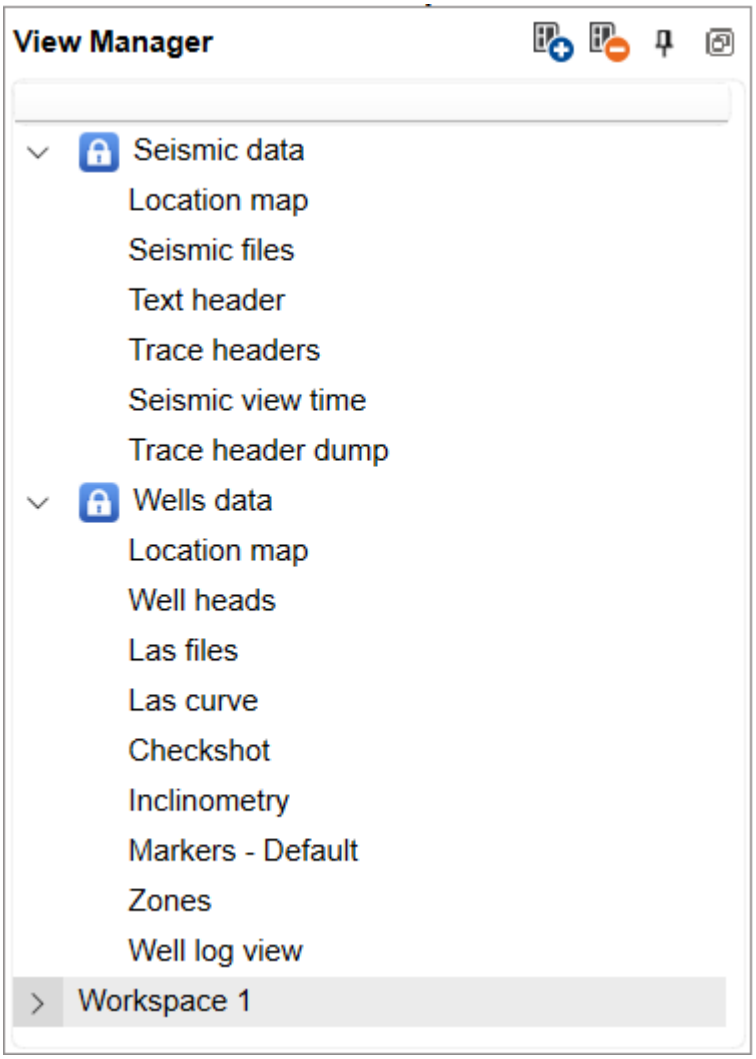
See Also

- Data manager 124

3.2.3 View Manager

View manager is a panel that allows to add, remove and manage different views 233 in numerous Workspaces 224 (equivalent to different desktops)

© 2026 Geomage 2003 LTD



Clicking a view or a workspace in the View Manager tree switches the application to the selected workspace. Right-clicking an item only moves the selection to it and opens the context menu for that item, while the workspace you are working in stays active. This lets you rename or remove another workspace or view without leaving your current workspace.
Views are displayed as tabs in the workspace area, and the active view is highlighted in blue. The cursor position can be synchronized across views, and each workspace remembers its own view layout.

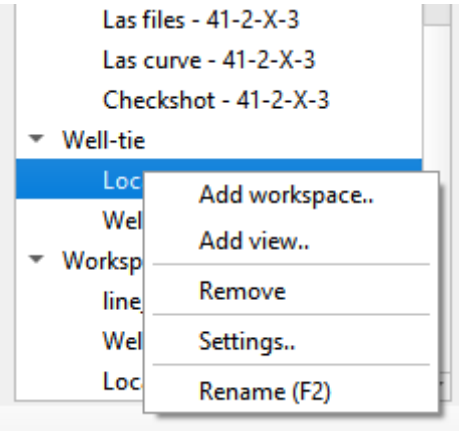
There are two predefined Workspaces 224 in the View Manager:
Seismic data 373 - this Workspace 224 has all views 233 required for importing 2D and 3D seismic into the project.
Wells data 391 - this Workspace 224 has all views 233 required for importing any well data to the project .

In the View Manager tree, both predefined Workspaces are marked with a lock icon next to their name. Workspaces created by the user are shown without this icon.

© 2026 Geomage 2003 LTD

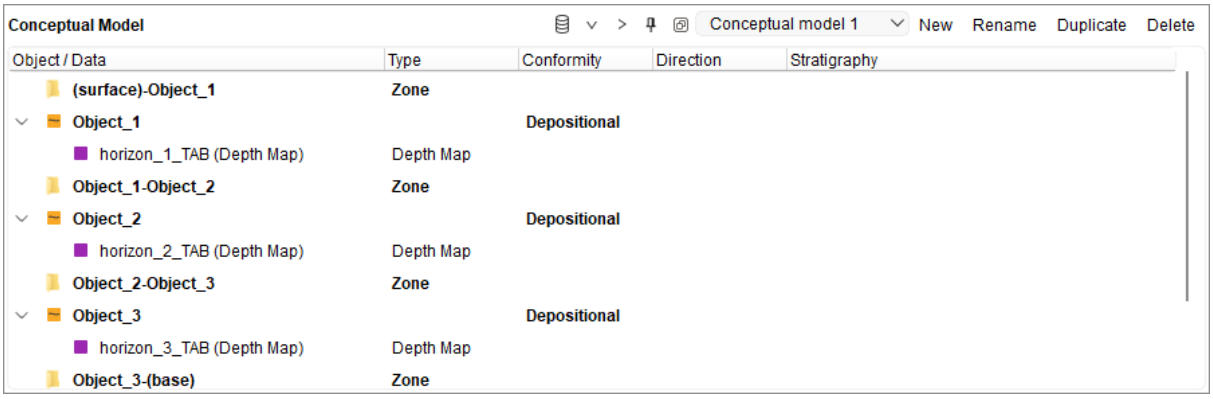
User cannot add, change or remove views from those predefined Workspaces.
User can change position and size of any view in those workspaces as well as graphical properties of any Object in those views.

User can change name of any workspace or view (except predefined) by either clicking F2 key or selecting Rename option from the context menu over desired view/workspace



3.2.4 Conceptual Model

The **Conceptual Model** panel is used to build a structured geological framework from existing project data. It does not store new data – instead it links objects from the Data Manager 124 to geological objects that can later be used in Structural Modeling 798.
The Conceptual Model is **domain-independent**: a single framework can reference both Time and Depth data at the same time. The domain is not chosen here – it is applied later, when the framework is used in Structural Modeling 798, where only the data matching the selected domain is used.
The panel is docked on the right side of the g-Space window, tabbed together with the Data Manager 124, **Object properties** and **Visual Settings** panels. (The View Manager 189 is docked separately, alone, on the left side of the window.)



© 2026 Geomage 2003 LTD

Conceptual Models in the Project

A project can hold several conceptual models, each with its own name and its own configuration. The combo box in the panel title bar selects the active model: the tree always shows the active conceptual model, and everything that uses the framework, such as Structural Modeling 798, works with the same active model. The actions next to the selector manage the models themselves:

- **New** – opens the **New conceptual model** prompt, where **Model name:** is pre-filled with the next free default name (**Conceptual model 1**, **Conceptual model 2**, and so on). The new model is empty and becomes the active one.
- **Rename** – opens the **Rename conceptual model** prompt for the active model.
- **Duplicate** – opens the **Duplicate conceptual model** prompt, where **New model name:** is pre-filled with the name of the active model followed by **(copy)**. The copy repeats the whole configuration of the active model – its objects, their assigned data, **Conformity**, **Direction** and **Stratigraphy** settings, its zones and its linked fault model – and becomes the active one.
- **Delete** – asks for confirmation and then removes the configuration of the active model only; the source data stays in the Data Manager 124. Deleting the last remaining model leaves a new empty **Conceptual model 1**.

Model names must be unique. If the entered name is already taken, g-Space reports it and the prompt stays open so that another name can be entered. A project saved before several conceptual models were supported opens as a single model named **Conceptual model 1**.

Geological Objects

Create and organize geological boundaries in top-to-bottom order. Each object represents a geological level (a horizon, an unconformity, or an intrusion) within the conceptual framework of the reservoir. Right-click in the tree to **Add Object**, **Add Object Above** or **Add Object Below** a selected object, **Rename** it, or **Remove Object**.

The tree is shown with the columns **Object / Data**, **Type**, **Conformity**, **Direction** and **Stratigraphy**. Object rows carry the Conformity and Direction values, while each linked data row shows its detected Type.

Linked Data

Drag objects from the Data Manager 124 and assign them to a geological object. Time and Depth maps, horizon points and horizon maps (Time or Depth), Multi-Z surfaces (Time or Depth), attribute maps, and Markers (shown as "Well Picks" in the panel) can all be linked, so that one geological object combines several data representations. The type and domain of each dropped item are detected automatically and shown in the **Type** column.

Each object may hold only one item per data type. Dropping an unsupported item type shows a warning and is ignored; dropping an item whose type is already present asks for confirmation before replacing the existing one. Items can also be linked without dragging: right-click an item in the Data Manager and choose **Assign to Conceptual Model** (or **Remove**

© 2026 Geomage 2003 LTD

from Conceptual Model). Right-click a linked data row and choose **Navigate to Data Manager** to jump back to the source item.

Conformity

Define the boundary type for each geological object:

- **Depositional** – a conformable boundary that follows normal deposition.
- **Unconformity** – an erosional or non-depositional surface that truncates older layers.
- **Intrusion** – a body that cross-cuts the surrounding stratigraphy.

Direction

For unconformities, define whether the object is **Top truncated** or **Bottom truncated**. This controls how the surface interacts with the layers above and below it during structural modeling. By default no direction is set.

Stratigraphy

Right-click a geological object and choose **Set Stratigraphy...** to associate it with a chronostratigraphic zone from the project's stratigraphy columns. The dialog shows one cascading combo box per stratigraphy column, from broadest to narrowest. Choose **Clear Stratigraphy** to remove the association.

When a marker is assigned to a geological object as **Well Picks**, g-Space compares its age with the project's current stratigraphy. If a matching zone is found and the object has no valid stratigraphy, g-Space fills the object's **Stratigraphy** value automatically. An existing valid assignment is preserved and can still be changed with **Set Stratigraphy...**.

Zones

When at least two geological objects take part in the layering, g-Space maintains the zones between them automatically. **Depositional** and **Unconformity** objects take part; **Intrusion** objects do not, and zones span across them. One zone is kept above the first participating object, one between each consecutive pair, and one below the last one, so that zone rows and object rows alternate from top to bottom.

A zone is named after the objects that bound it – for example, **Object_1-Object_2** – and the open ends of the column are shown as **(surface)** and **(base)**. A zone row shows **Zone** in the **Type** column, leaves **Conformity** empty and inherits the **Stratigraphy** of its upper bounding object. If the ages of the two bounding objects are out of column order, the inherited value is marked **(check order)**.

A zone holds one **Zone Markers** item and one **Isochore Map**, together with any number of polygons and property maps. Drag the items from the Data Manager 124 onto the zone row: a project zone is taken as the zone markers, a depth map as the isochore map, and other maps and polygons are added to the zone. The same assignments are available by right-clicking the zone – **Assign Zone Markers...**, **Assign Isochore Map...**, **Add Polygon...** and **Add Property Map...** – and when the project already contains a zone with the same name, the menu offers it

© 2026 Geomage 2003 LTD

directly as **Assign Suggested Zone**. Choose **Rename Zone** to give the zone your own name. An item that a zone cannot hold is refused with an **Invalid zone assignment** message.
Only assigned items are listed under a zone, so an empty zone stays a single row, and a zone is expanded automatically after an assignment.
Adding, removing or reordering an object, and switching its **Conformity** to or from **Intrusion**, recalculates the neighboring zones as one step that can be undone at once. Renamed zones and their assigned items survive such a recalculation; when a change would destroy a renamed zone or a zone with assigned data, a **Zone destruction** confirmation lists what would be lost before the change is applied.
A warning icon in place of the zone icon means that one of the two objects bounding the zone has unresolved inputs – no data assigned, an **Unconformity** with no **Direction**, or an assigned item that no longer resolves in the Data Manager 124, such as a Multi-Z surface whose source item was removed; point at the row in the **Object / Data** column to read **A bounding top of this zone has unresolved errors**. While a zone is in this state, its assigned **Zone Markers** row shows **— (top has errors)** instead of the name of the item.

Fault Model

Optionally link one global **Fault Model** to the conceptual model. Only one fault model can be linked; assign it by right-clicking a fault model in the Data Manager 124 and choosing **Assign as Conceptual Model fault model**. When present, this fault model is preselected as the default in the Structural Modeling 798 wizard, where the fault framework is taken into account when the structural model is built.

Clearing the Model

Right-click an empty area of the panel to clean up the active conceptual model:

- **Clear all** – removes the whole configuration of the active conceptual model, including every geological object with its assigned data, the zones between them and the linked **Fault Model**, and returns the panel to its empty state.
- **Delete assigned data for all objects** – removes the assigned data from every geological object but keeps the objects themselves, with their names and their **Conformity**, **Direction** and **Stratigraphy** settings.

Both actions ask for confirmation before they run. To clear a single object, right-click it and choose **Delete assigned data**: only the data assigned to that object is removed and the other objects stay unchanged.
Each action appears only when there is something for it to remove. All three affect the conceptual model only – the source items are not deleted from the Data Manager 124.

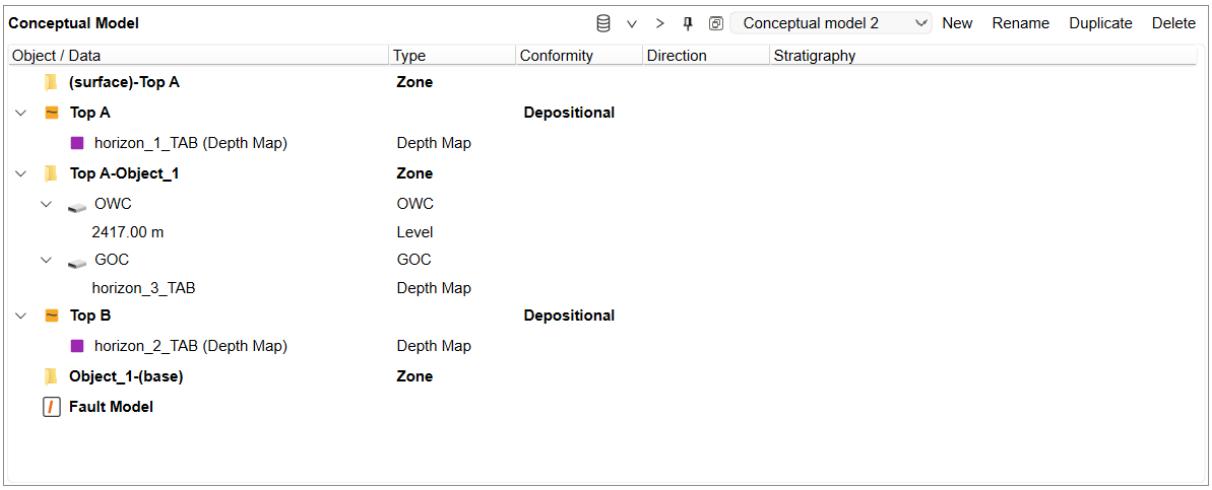
See Also

Data Manager 124
View Manager 189
Structural modeling 798
Panels 123

© 2026 Geomage 2003 LTD

3.2.4.1 Fluid contacts

A fluid contact records the boundary between two fluids inside one zone of the Conceptual Model 191: an oil-water contact (**OWC**), a gas-oil contact (**GOC**), a gas-water contact (**GWC**) or the free water level (**FWL**). Contacts are added and edited only in the **Conceptual Model** panel, and they are informational – they are not used in structural modeling or in any calculation.
A zone is the interval between two adjacent geological objects and is shown as a **Zone** row in the tree. Contacts are added to a zone and appear as separate rows beneath it, with the contact type in the **Type** column. Each assignment that defines a contact is listed as a child row below the contact.



Adding a contact

1. Right-click a **Zone** row and open the **Add Contact** submenu. On a geological object row, use **Add Contact ▸ Zone Above** or **Add Contact ▸ Zone Below** instead.
2. Choose the type of the contact: **OWC**, **GOC**, **GWC** or **FWL**.

The contact is created under the zone and takes its type as name; a second contact of the same type is named **OWC (2)**, and so on. A zone can hold as many contacts as needed – for example one contact per fault block. Right-click a contact and choose **Rename** to edit its name in place, or **Remove Contact** to delete it.

Defining a contact

A contact is defined either by a constant level, optionally limited laterally by a polygon, or by a surface. Right-click the contact and use:

- **Set Level...** – enter the depth of the contact in the **Level (depth)** field of the dialog that opens.
- **Assign Polygon...** – choose a polygon that limits the contact laterally, for example to one fault block.

© 2026 Geomage 2003 LTD

- **Assign Map...** – choose a depth map that the contact follows, for a tilted or variable contact.

The two options are mutually exclusive: assigning a map clears the level and the polygon, and setting a level or assigning a polygon clears the map. A depth map or a polygon can also be dragged from the Data Manager 124 onto the contact row; other item types are refused with a message.
Each assignment is listed as a child row of the contact, with **Level**, **Polygon** or **Depth Map** in the **Type** column; a level row shows the depth with its unit. Right-click such a row and choose **Change...** to select a different value, or **Remove** to clear it. Maps and polygons are references: the contact points at the item in the Data Manager and does not copy it.

Warnings

A warning icon on a contact row means that the contact needs attention. Point at the row to read the reason:

- No data is assigned yet, so the contact has nothing to draw.
- A referenced Data Manager item is missing, for example because the map or the polygon was deleted. The contact itself is kept, so a new item can be assigned to it.
- The zone changed. When a geological object is removed, the zones above and below it merge, and the contacts of the affected zones move to the merged zone instead of being deleted – check that their definition is still valid.

Showing contacts in the Data Manager

Switch on **Show in Data Manager** on the **Conceptual Model** panel title bar to list the contacts in the Data Manager 124. A **Fluid contacts** folder appears, with one subfolder per zone and one entry per contact, named after its type and its name. Use the check box of an entry to show or hide that contact in the views. A contact with no data assigned has nothing to draw, so its check box is disabled. The folder is a view of the Conceptual Model and stays in step with it; contacts are still added, renamed and defined in the **Conceptual Model** panel.

Contacts on sections

A visible contact is drawn as a horizontal line on seismic sections – inline, crossline and arbitrary, in both the time and the depth domain – and on Well correlation view 296 sections, at the depth where the contact crosses each well trajectory.
A contact defined by a level is drawn at that constant depth, and a contact defined by a map follows the map along the section. Where a polygon is assigned, the line is drawn inside the polygon only and breaks outside it. Contacts are defined in depth, so on a time section the line is converted with the project velocity model.
Each contact type has a default colour: OWC blue, GOC green, GWC teal, and FWL blue with a dashed line. In Visual Settings 206 the section overlay is listed as **All fluid contacts**, with one entry per contact named after its type and its level or map – **OWC 2417.0**, for example. The colour and the line style of each entry can be changed there.

© 2026 Geomage 2003 LTD

See Also

Conceptual Model 191
Data Manager 124
Well correlation view 296
Visual Settings 206

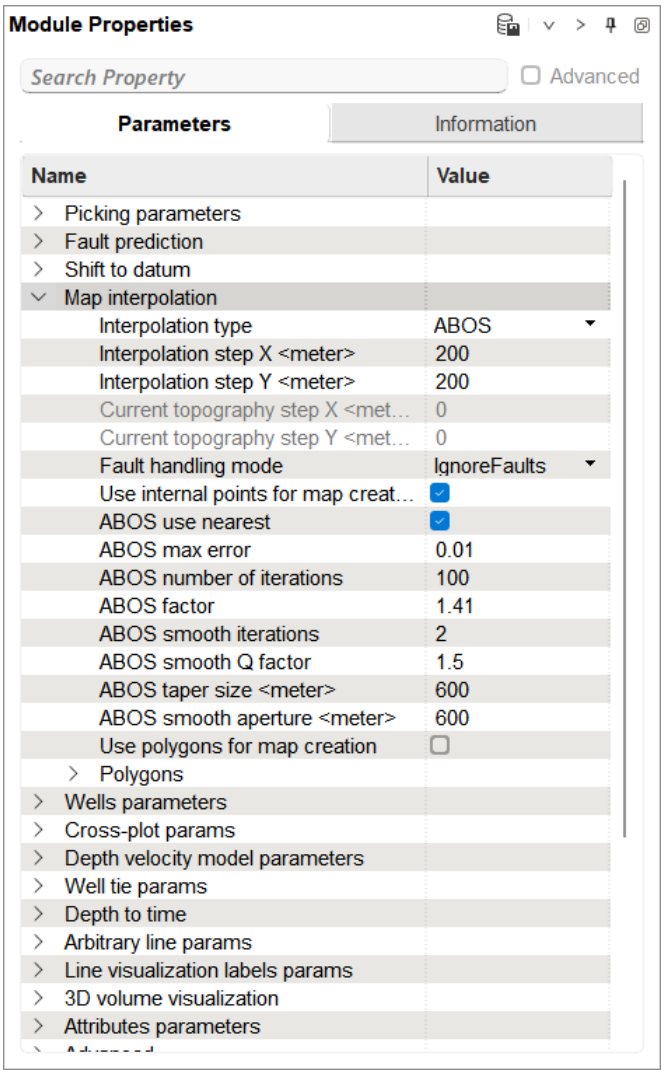
3.2.5 Module Properties

Module properties allows to setup all the process parameters implemented in g-Space Changes made in Module Properties affect the active view immediately.

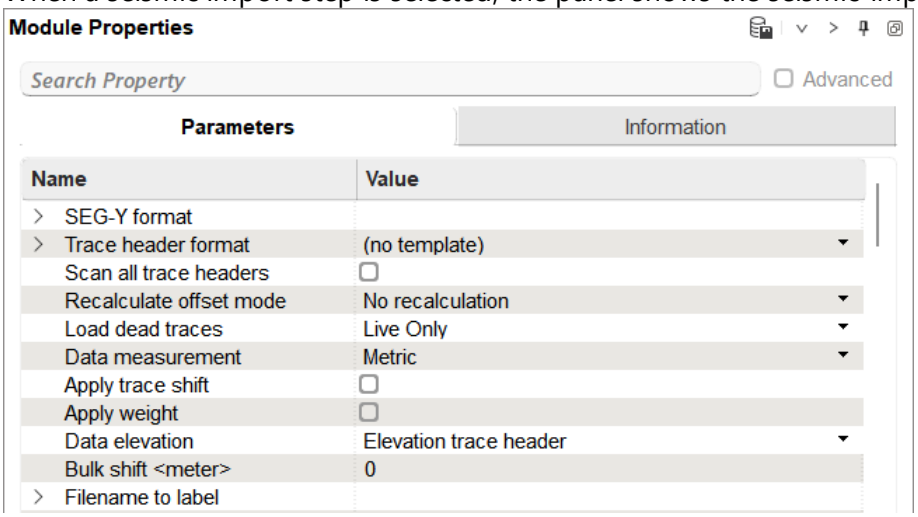
Module Properties always shows the parameters of whichever module or procedure is currently selected in the flow, split into a **Parameters** tab and an **Information** (output) tab. Its content changes for every module — for example, selecting a SEG-Y import step shows the seismic-import parameters (below), while wells, horizons, velocity and attributes modules show their own parameter sets. Modules such as reservoir volume / HCIIP calculation and Python scripting also expose their own parameters in this panel.
A **Search Property** box at the top of the panel filters the visible parameters by name.

When the top-level **g-Space** item is selected, the panel shows project-wide process parameters. The example below expands **Map interpolation**:

© 2026 Geomage 2003 LTD



When a seismic import step is selected, the panel shows the seismic-import parameters:



© 2026 Geomage 2003 LTD

List of properties in default state:

Legacy selection parameters. The **Current layer** and **Current fault** entries are no longer shown in Module Properties. Select the current layer on the **Interpretation** 96 ribbon tab; the former **Current fault** parameter is obsolete.

Function Name	Explanation
Cross-correlation length <ms>	Sets the length of the cross-correlation window in milliseconds.
Distance for X traces <Distance>	Defines the distance used for cross-correlation across traces.
Shift from keyboard <ms>	Allows manual input of shift values in milliseconds via the keyboard.
Postfix	Adds a postfix to the cross-correlation results.
Intersections projection radius <feet>	Sets the projection radius for intersections
Replacement velocity <ft/s>	Specifies the replacement velocity in m/ft per second
Import measurement system	Sets the measurement system on import
Shift to datum	
Datum time <feet>	Sets the datum for data in time domain
Datum depth <feet>	Sets the datum for data in depth domain
Manual picking params	Opens parameters for manual picking settings
Pick mode	Sets the mode for picking (tracking, interpolation, etc.)
Automatic picking window <ms>	Defines the window for automatic picking in milliseconds
Pick type	Specifies the type of picking (e.g., peak, trough, zero-crossing)
Magnet type	Defines the magnet type for picking assistance
Energy picking threshold <%>	Sets the energy threshold for automatic picking in percentage
Horizon track on bin change	Enables horizon tracking on bin changes
Horizon Tracking	
Inline/cdp start autopick	Sets the starting inline/CDP for automatic picking
Crossline start autopick	Sets the starting crossline for automatic picking
Inline/cdp end autopick	Sets the ending inline/CDP for automatic picking
Crossline end autopick	Sets the ending crossline for automatic picking
Inline/cdp step autopick	Defines the step size for inlines/CDPs during automatic picking
Crossline step autopick	Defines the step size for crosslines during automatic picking
Follow IL XL continuity	Ensures continuity is followed along inlines and crosslines

© 2026 Geomage 2003 LTD

By corr	Uses correlation for horizon tracking
Map Interpolation	
Interpolation type	Sets the type of interpolation used for map creation
Interpolation step X <feet>	Defines the interpolation step size in X direction
Interpolation step Y <feet>	Defines the interpolation step size in Y direction
Current topography step X <feet>	Defines the current topography step size in X direction
Current topography step Y <feet>	Defines the current topography step size in Y direction
Fault handling mode	Controls how faults are used during map creation
Use internal points for map creation	Uses internal points for map creation
Fault clear radius <feet>	Sets the map clearing radius around faults
Kriging covariance	Defines the covariance for Kriging interpolation
Kriging range <feet>	Sets the range for Kriging interpolation
Kriging max points	Defines the maximum number of points for Kriging interpolation
Use polygons for map creation	Includes the bounding polygon in the map creation process
Polygons	Manages the polygon used for map creation
Wells Parameters	
Well projection radius <feet>	Sets the projection radius for wells
Well labels overlapping protection	Enables protection to prevent well labels from overlapping
Synthetic seis width <Trace>	Sets the width of synthetic seismic traces
Nearest seis type	Specifies the type of nearest seismic data used (distance or number of traces)
Nearest seis width <Trace>	Sets the width of the nearest seismic traces
Nearest seis distance <feet>	Defines the distance for the nearest seismic data
Selected seismic	Sets the currently selected seismic data
Selected seismic group	Sets the currently selected seismic group
Synthetic seismogram	
Create	Sets the data for a synthetic trace generation (sonic log, sonic - density, etc.)
Sonic	Sets the sonic log data for synthetic seismogram creation
Density	Sets the density log data for synthetic seismogram creation
Impedance	Sets the impedance log data for synthetic seismogram creation
Use anti-aliasing filter	Applies an anti-aliasing filter to the logs data
Update impedance on change checkshot	Updates impedance when checkshot data changes
Depth smooth <feet>	Sets smoothing step to depth data
Depth resample <feet>	Sets resampling step to depth data

© 2026 Geomage 2003 LTD

Single well view template	Defines the template for single well view
Markers colors from stratigraphy	Uses stratigraphy to color well markers
Stratigraphy column	Specifies the stratigraphy column used for markers
LAS for Well color	Sets the LAS to color wellbores in 3D view
LAS for Well width	Sets the LAS for wellbores width in 3D view
LAS curve step	Defines the step size for LAS curves
Cross-plot Params	
Cross plot markers name	Sets the marker to cross-plot
Cross plot horizon name	Sets the horizon to cross-plot
Depth Velocity Model Parameters	
Datum <feet>	Sets the datum level
Max depth <feet>	Defines the maximum depth for the model
Depth step <feet>	Sets the step size for depth
Bottom velocity <ft/s>	Defines the bottom velocity
Velocity for constant model <ft/s>	Sets the velocity for a constant model
Create depth velocity from volume params	
Velocity type	Sets the type of velocity stored in the input volume: RMS, interval or average
Vertical smoothing window <Samples>	Sets the vertical smoothing window for the created velocity model in samples
Horizontal smoothing window <Traces>	Sets the horizontal smoothing window for the created velocity model in traces
Create constant layer depth velocity params	
Number of supersamples	Defines the number of supersamples for constant layer depth velocity creation
Well Tie Params	
Bottom taper <ms>	Sets the bottom taper
Max radius to well <feet>	Defines the maximum radius to well
Propagation taper min <feet>	Sets the minimum taper for propagation
Propagation taper max <feet>	Sets the maximum taper for propagation
Max velocity <ft/s>	Defines the maximum velocity
Min velocity <ft/s>	Defines the minimum velocity
Gradient type	Sets the gradient type for well tie
Checkshot welltie	
Max marker deviation from well <feet>	Sets the maximum deviation for markers from the well
Interpolation type	Defines the interpolation type for checkshot welltie
Trend/Voronoi/triangulation smooth <feet>	Sets the smoothing type and range for trend, Voronoi, and triangulation.
Kriging covariance	Defines the covariance for Kriging in welltie

© 2026 Geomage 2003 LTD

Kriging range	Sets the range for Kriging in welltie
Marker type interpolation	Specifies the interpolation type for markers
Regression coeff	Sets the regression coefficient for welltie
Extrapolate checkshot	Enables extrapolation of checkshot data
(Checkshot + marker + horizon) Calculate velocity by checkshots	Calculates velocity using checkshot, marker, and horizon data
(Checkshot + marker + horizon) Add marker horizons	Adds marker horizons to the checkshot and marker data
(Checkshot + marker + horizon) Vrms creation step	Sets the step size for Vrms creation from checkshot, marker, and horizon data
Extrapolate deepest layer coeff	Sets the coefficient for extrapolating the deepest layer
Max distance to layer <feet>	Defines the maximum distance to the layer
Use wells and horizons inside polygon	Uses wells and horizons within a specified polygon for calculations
Use marker projection for detect velocity if marker not exist	Projects marker data to detect velocity if the marker is absent
Calculate velocity from topography	Computes velocity starting from topography level
Extrapolate check shot to 0	Extrapolates check shot data to zero
Depth to Time	
DT <ms>	Sets the depth to time conversion factor in milliseconds
Max time <ms>	Defines the maximum time for depth to time conversion
Arbitrary Line Params	
Create arbitrary line	Creates an arbitrary line
Arbitrary line step <feet>	Sets the step size for the arbitrary line
Arbitrary line max search distance <feet>	Defines the maximum search distance for the arbitrary line
Straighten on 3D view	Straightens the arbitrary line in the 3D view
Line Visualization Labels	
Show	
Trace header	
Show header name	
Precision	Sets the precision for line visualization labels.
Step <feet>	
3D Volume Visualization	
Step inline	Defines the step size for inlines in 3D volume visualization
Step crossline	Defines the step size for crosslines in 3D volume visualization
Step time <ms>	Sets the time step size in milliseconds for 3D volume visualization

© 2026 Geomage 2003 LTD

Step depth <feet>	Sets the depth step size in feet for 3D volume visualization
Current inline	Sets the current inline being visualized
Current crossline	Sets the current crossline being visualized
Current time <ms>	Sets the current time for slice visualization
Current depth <feet>	Sets the current depth for slice visualization
Attributes Parameters	
Amplitude	Computes the amplitude attribute
Envelope	Computes the envelope attribute
Instantaneous phase	Computes the instantaneous phase attribute
Instantaneous unwrap phase	Computes the unwrapped instantaneous phase attribute
Instantaneous frequency	Computes the instantaneous frequency attribute
Frequency derivative	Computes the frequency derivative attribute
Instantaneous bandwidth	Computes the instantaneous bandwidth attribute
Cosine of instantaneous phase	Computes the cosine of the instantaneous phase attribute
Dominate frequency	Computes the dominant frequency attribute
Thin bed indicator	Computes the thin bed indicator attribute
Envelope derivative	Computes the derivative of the envelope attribute
Second derivative of envelope	Computes the second derivative of the envelope attribute
Instantaneous Q	Computes the instantaneous Q attribute
Coherence	Computes the coherence attribute
Chaos	Computes the chaos attribute
Seismic relief	Computes the seismic relief attribute
Check/uncheck all attributes	Toggles the selection of all attributes for calculation
Selected horizon	Sets the horizon selected for attribute calculation
Time window <ms>	Sets the time window for attribute calculation
Calc type	Defines the type of calculation for attributes (map or the entire volume)
Advanced	
Merge stack line distance <feet>	Sets the distance for merging seismic volumes/lines into the single survey
Max distance picking projection <feet>	Defines the maximum distance for picking projection
Turn on undo	Enables the undo feature
Fill gaps in volume	Fills gaps in the volume data
Performance	
Number of threads	Sets the number of processing threads (default N-1)
Show well names on location map	Displays well names on the location map

© 2026 Geomage 2003 LTD

Show las curves on seismic views	Displays LAS curves on seismic views
Horizon performance	
Update horizon phase maps	Updates the horizon phase maps
Update horizons 3d view	Updates horizons in the 3D view
Update horizons picking 3d view	Updates horizon picking in the 3D view
Update volume 3D view	Updates the volume in the 3D view
Update maps and faults after adding stick	Updates maps and faults after adding a stick
Use only live traces for 3D data contouring	Uses only live traces to create the volume polygon
Show well table	Displays the well table
AVO real time update on polygons change	Updates AVO data in real time on polygons
Show tet grids	Displays tetrahedral grids
3D autopicking update GUI bulk	Updates the GUI in bulk during 3D autopicking
Wavelet	
Convert to min phase	Converts the wavelet to minimum phase
Wavelet length <ms>	Sets the length of the wavelet
Wavelet Sample rate <ms>	Sets the sample rate of the wavelet
Ricker type	Specifies the type of Ricker wavelet
Dominant frequency <Hz>	Sets the dominant frequency of the wavelet
Detect time start <ms>	Defines the start time for wavelet detection
Detect time end <ms>	Defines the end time for wavelet detection
Fault Prediction	
Correlation time window <ms>	Sets the time window for cross correlation
Correlation bin window	Sets the bin window for cross correlation
Maximum shift <ms>	Defines the maximum shift allowed in fault prediction
Time stretch <%>	Specifies the time stretch percentage for fault prediction
Sigma for local faults	Sets the sigma value for local fault detection
Sigma for generalized faults	Sets the sigma value for generalized fault detection
Sigma for noise reduction	Sets the sigma value for noise reduction
Autopicking step	Defines the step size for autopicking
Coordinate Converting	
Use converting on import	Applies coordinate conversion on data import
Reinterpolation Params	
Do reinterpolation	Enables reinterpolation of data
Reinterpolation radius <feet>	Sets the radius for reinterpolation in feet
Number of taken bins	Defines the number of bins taken for reinterpolation
Draw bulk on import shape files	Draws bulk data on import of shape files

© 2026 Geomage 2003 LTD

List of seismic import parameters (Seismic data workspace):

Function Name	Explanation
SEG-Y Format	
Data type	Specifies the type of data in the SEG-Y format (land, marine, etc.)
Text header size	Defines the size of the text header in the SEG-Y file
Binary header size	Defines the size of the binary header in the SEG-Y file
Endian	Specifies the byte order (endianness) of the data
EBCDIC	Indicates whether the text header is in EBCDIC format
Data Sample format	Specifies the format of the data samples in the SEG-Y file (int, float, etc.)
Trace Header Format	Sets the trace header format from the list
Scan all trace headers	Scans all trace headers when reading the file
Recalculate offset mode	Sets the mode for recalculating offsets in the trace headers
Load dead traces	Option to load or not dead traces from the SEG-Y file
Data measurement	Specifies the measurement units for the data
Apply trace shift	Applies a shift to trace data
Apply weight	Applies a weight factor to trace data
Data datum	Sets the datum level for the data (e.g. constant, floating)
Data elevation	Sets the topography level for the data
Constant datum <feet>	Specifies a constant datum level
VO <ft/s>	Specifies the replacement velocity (VO)
Bulk shift <ms> / <meter>	Applies a bulk shift to the data. The unit follows the Domain set for the file: the project time unit for time-domain data, the project depth unit for depth-domain data
Filename to Label	
First char	Specifies the first character position used for labeling
Number of chars	Defines the number of characters used for labeling
Label	Displays the current label
Domain type	Specifies the domain of the data (time or depth)
Data type	Specifies the type of data (amplitude, velocity, etc.)
Options	Defines data type between pre-stack and post-stack
Group	Specifies the data group (e.g. PSTM, PSDM)
Data dimension	Defines the dimension of the data (2D or 3D)
Load to RAM	Option to load data into RAM
State	Displays the state of loading data into the project
Update topography	Updates the topography data after loading SEG-Y

© 2026 Geomage 2003 LTD

Number of traces for preview	Specifies the number of traces to preview
File name	Displays the name of the file being processed
Number of bins along inline	Displays the number of bins along the inline direction
Number of bins along crossline	Displays the number of bins along the crossline direction
Inline step	Displays the step size for inlines
Xline step	Displays the step size for crosslines
Distance Inline <feet>	Displays the distance between inlines
Distance crossline <feet>	Displays the distance between crosslines
Number of CMP	Displays the number of common midpoints (CMP) in the data
Inline azimuth <degree>	Displays the azimuth angle for inlines
Minimum offset <feet>	Displays the minimum offset distance
Maximum offset <feet>	Displays the maximum offset distance
From azimuth <degree>	Displays the starting azimuth angle
To azimuth <degree>	Displays the ending azimuth angle

See Also

Panels 123
User Interface 30
Ribbon bar 33

3.2.6 Visual Settings

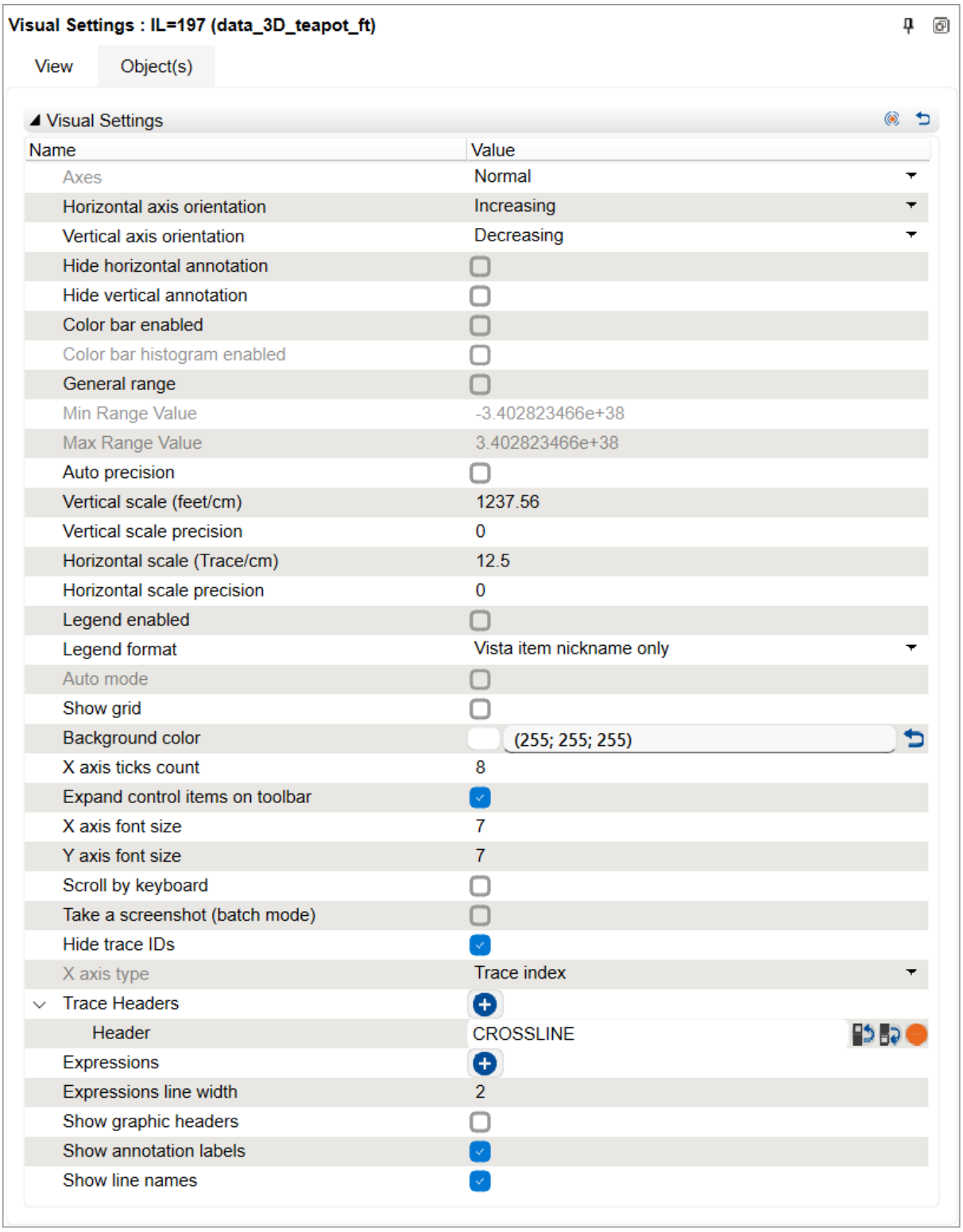
Visual Settings is a dynamic panel that displays the parameters for the active view or object. The content of the window depends of the selected object.
Visual Settings changes only the display style of the active view and the objects shown in it – it does not change the original data stored in the project.

By default Visual Settings is located on the right.

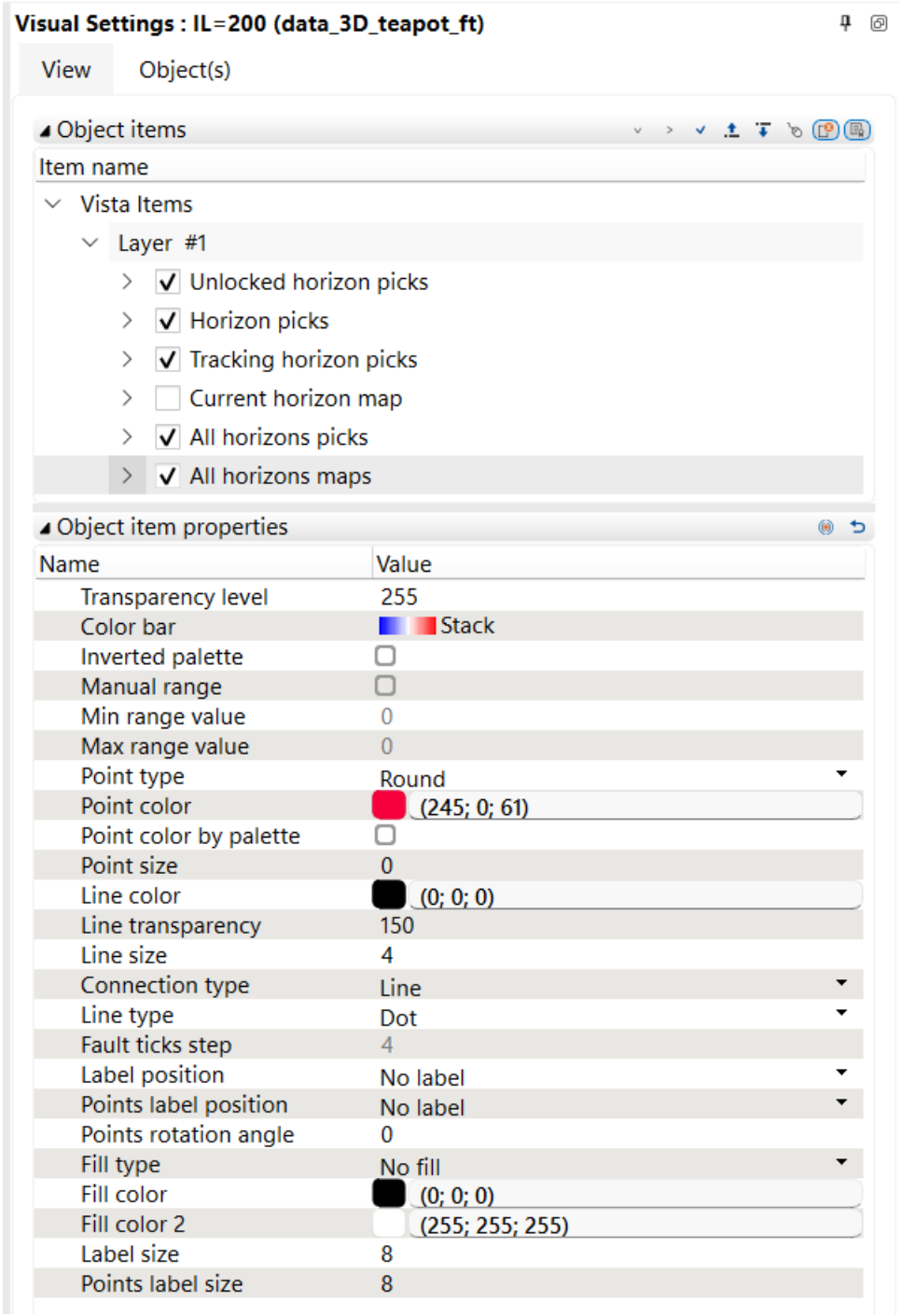
To open the Visual Settings panel open a panel bar or click the Settings button on the tool bar on the upper part of any View.

Visual Settings is divided to two tabs:
View 208 - controls the appearance of the active view, including axes, scale, annotations, grid, legend, background colour, headers and display options.
Object 210 - controls the objects displayed in the active view, including their visibility, order, palette, transparency, point style, line style, labels and fill settings.

© 2026 Geomage 2003 LTD



© 2026 Geomage 2003 LTD

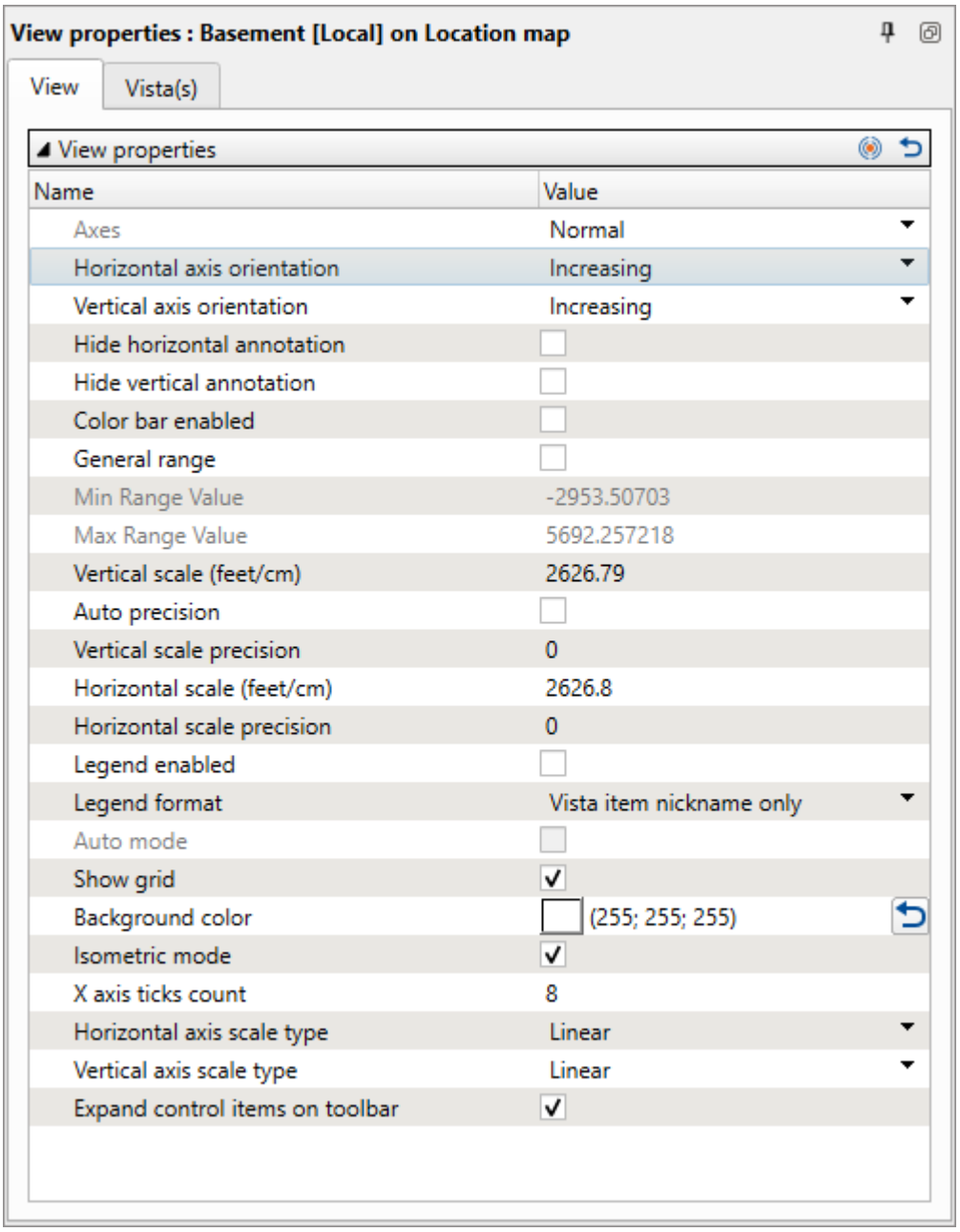


3.2.6.1 View

View tab - allows user to adjust views visual setting in g-Space. The settings list may vary depending on the active view type.

© 2026 Geomage 2003 LTD

The View tab works together with the Object 210 tab of the Visual Settings panel, which controls the visibility and appearance of the data objects displayed in the active view.



In the upper part you can reset properties to default parameters or propagate properties to other same type views

Here is the list of the settings you can make for the active view (it may not be complete and depends on the active view type)**:**

© 2026 Geomage 2003 LTD

- Axes - default: Normal, but currently that property is disabled. Designed to change Horizontal and Vertical axes by rotating it 90 degrees.
- Horizontal/vertical axis orientation - flips orientation of horizontal/vertical axis
- Hide horizontal/vertical annotation - hide annotation from horizontal/vertical axis
- Color bar enabled - when enabled adds color bar on right side of the view
- General range - when selected, the program will consider next 2 parameters for colorbar range of the pallete
- Min/Max Range Value - when enabled define minimum/maximum value of the palette for all Objects on the view
- Vertical scale (feet/cm) - defines the actual vertical scale of the view
- Auto precision - when checked, application will define automatically the precision of vertical scale
- Vertical scale precision - when auto-precision not checked, this parameter defines precision for the vertical scale
- Horizontal scale (feet/cm) - defines the actual horizontal scale of the view
- Horizontal scale precision - when auto-precision not checked, this parameter defines precision for the horizontal scale
- Legend enabled - when enabled adds legend on the view
- Legend format - defines legend format
- Auto mode - default settings
- Show grid - shows grid on the view
- Background color - defines background color
- Isometric mode - when checked, vertical and horizontal scales will be the same
- X axis ticks count - number of ticks for horizontal axis
- Horizontal axis scale type - defines linear or log type for horizontal scale
- Vertical axis scale type - defines linear or log type for vertical scale
- Expand control items on toolbar - allows to show more icons in the upper part of the view
- Hide trace IDs - hides traces IDs on section views
- Trace Headers - allows to add additional information to section views from the segy headers
- Show line names - shows the names of the 2D seismic lines that the section crosses in the annotation area above it; on an arbitrary line, every 2D line the path follows is named. Selected by default. Clear it to hide the line names and keep the other annotation rows, which are switched as a whole by Show annotation labels

3.2.6.2 Object

The **Object(s)** tab lets you set the visibility, order, and appearance of all objects in the active view.
It is divided into two sections: **Object items** and **Object item properties**

© 2026 Geomage 2003 LTD

Object items - allows you to turn on/off visibility of the objects on the view. This is dynamic part of the window and shows the items according to the active view

You can re-order the layers , turn on/off visibility of all objects and switch between working modes on the current view. The available Object items depend on the view and loaded data and can include **Fault picks**, **Horizon picks**, **2D Seismic time**, **Converted 2D Seismic time**, **2D Seismic depth**, **Converted 2D Seismic depth**, **Inline time**, **Crossline time**, **Inline depth**, or **Crossline depth**.

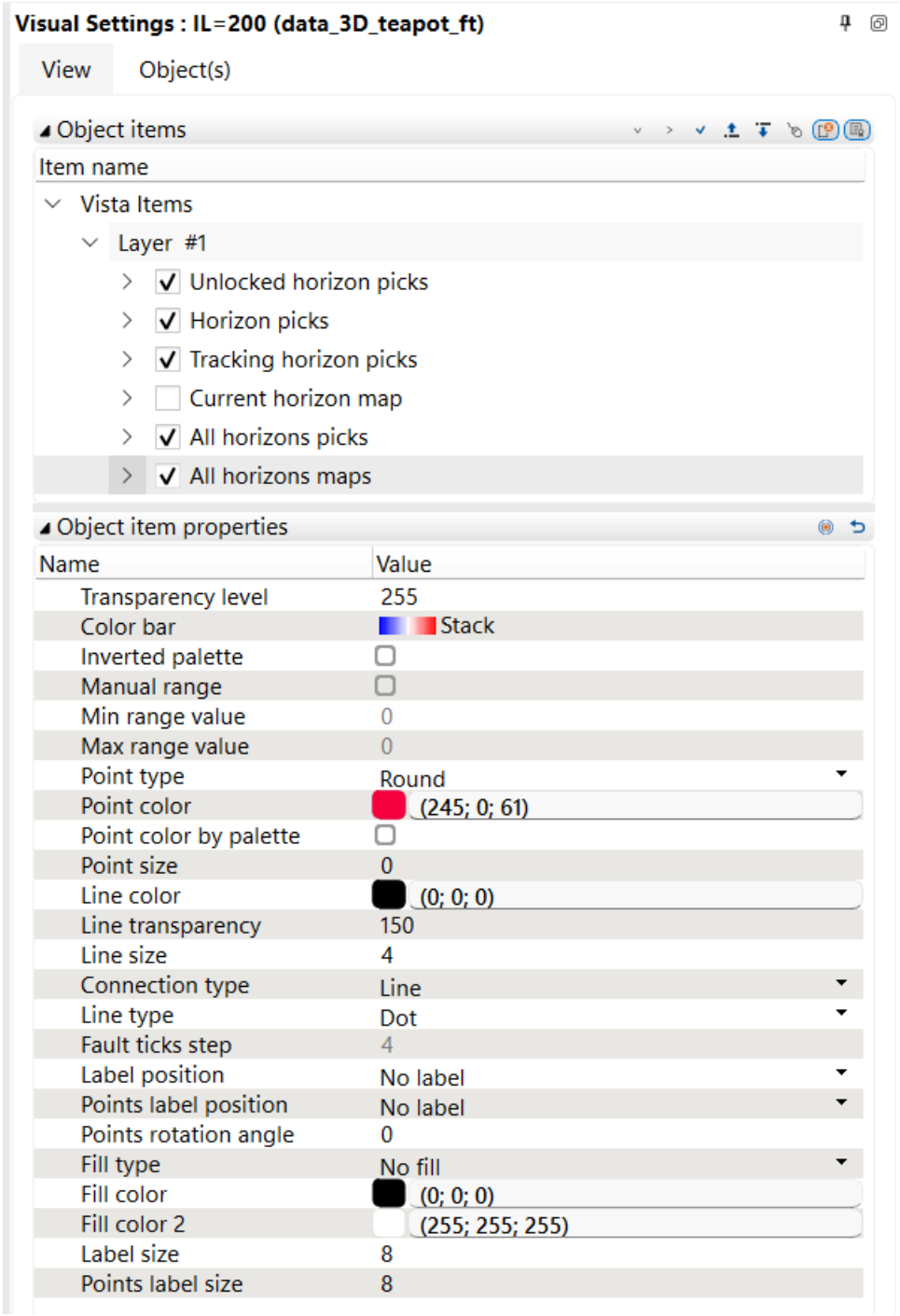
The **Hide empty objects** toggle in the Object items header (turned on by default) hides objects that have no items on the current view.

Additionally you can filter Object items by current object type in data manager (turned on by default). To do so here the general flow in few steps:

1. Open Visual Settings panel (by opening the panel bar or using button)
2. Select **Filter by Data manager** .
3. In **Data Manager**, select the type of object you want to manage.

After that in the Visual Settings you will get Object items for only the objects you selected in Data Manager:

© 2026 Geomage 2003 LTD



Object item properties section gives access to settings of selected object in the **Object items** section.
The settings list may vary depending on the active item type

© 2026 Geomage 2003 LTD

Here is the list of the settings you can make for the active item (it may not be complete and depends on the active item type)**:**

- Transparency level - set the opacity level of the item
- Color bar - allows to select a color bar
- Inverted palette - inverts a color bar
- Normalization - allows to select the source for color range
- Manual range - enables user-defined minimum and maximum range values
- Min range value - sets the lower value for user-defined normalization
- Max range value - sets the upper value for user-defined normalization
- Hide out of range - hides values outside the selected range values
- Gain - allows to choose gain for the visualization
- AGC <ms> - allows to set AGC
- Density - displays the item as a variable-density plot
- Wiggle - displays the item as wiggle traces
- Interpolated values type - sets how matrix values are interpolated for display: None, Bilinear (default) or Smooth
- Enable errors on matrix - enables display of error values stored with the matrix
- Auto normalization - automatically sets the color range from the data
- Value precision - set precision of the displayed data
- Isolines show mode - allow to turn on isolines
- Major each isoline - set the step of the bold isolines
- Minor line size - set thickness of minor isolines
- Minor line color - set color of minor isolines
- Major line size - set thickness of major isolines
- Major line color - set color of major isolines
- Isolines round mode - set isolines smoothing
- Show isolines labels - shows isolines values
- Isolines color by palette - set isolines color according the color bar
- Point type - set points style
- Point color - set points color
- Point color by palette - set points color according the color bar
- Point size - set points size
- Line color - set line color
- Line transparency - set opacity level for the line
- Connection type - set connection type for the points
- Line type - set line style (like solid, dashed, dotted etc.)
- Label position - set the label position
- Points label position - set the label position for the points
- Points rotation angle - set the points rotation
- Fill type - allow to fill the object
- Fill color - set the first filling color

© 2026 Geomage 2003 LTD

- Fill color 2 - sets the second fill color

Color range of 3D seismic sections

When an inline or crossline section of a 3D seismic amplitude or attribute volume is shown in a view, its color range is taken from the whole volume and not from the current section. For such an item **Normalization** is set to **User Defined**, **Manual range** is turned on, and **Min range value** and **Max range value** hold the minimum and maximum of the entire volume. Amplitudes therefore stay comparable when you move from one section to another and between inline and crossline sections.
This range is set when the section is first displayed. If you afterwards choose your own **Normalization** or range values, they are kept when you move to another section.

Setting the symbol of a single well

A collection listed in the **Object items** section can be expanded to the individual objects it contains - for example the separate wells inside **Wells on map**. Select one of these individual items to edit its own settings in the **Object item properties** section.
To give one well its own symbol, expand the collection, select the well, and set the required **Point type**. A symbol set this way belongs to that well alone: it is kept when the symbol of the collection or group the well belongs to is changed afterwards, and it is stored with the project.

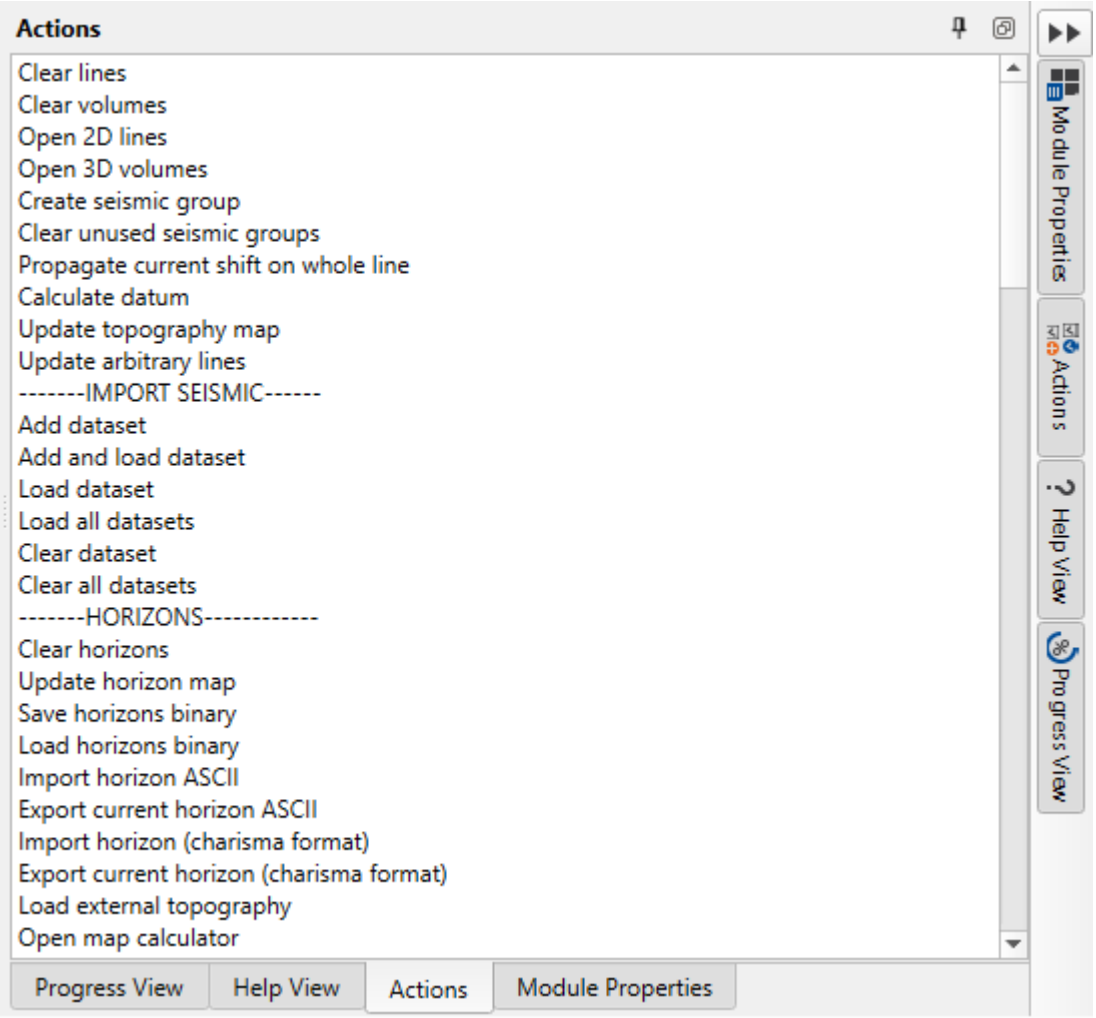
See Also

Visual Settings 206
View 208
Panels 123

3.2.7 Actions

Actions panel allows to see all available processes that can be executed in g-Space

© 2026 Geomage 2003 LTD



All these actions are mostly duplicated by another button, toolbar or tool, but some of them can still only be found in the Actions panel, especially new features.

Function Name	Explanation
SEISMIC DATA	
Clear lines	Removes all loaded 2D seismic lines from the project
Clear volumes	Removes all loaded 3D seismic volumes from the project
Open 2D lines	Navigate 2D seismic lines for loading into the project
Open 3D volumes	Navigate 3D seismic volumes for loading into the project
Create seismic group	Creates a new group to organize multiple seismic datasets (for example PSTM, PostSTM etc.)
Clear unused seismic groups	Deletes seismic groups that are not currently in use
Propagate current shift on whole line	Applies the current time or depth shift across the entire seismic line
Calculate datum	Computes the reference level or datum for seismic data

© 2026 Geomage 2003 LTD

Update topography map	Refreshes the topography map with the latest seismic data
Update arbitrary lines	Updates user-defined arbitrary seismic lines with the latest data
Create 3D geological model	Generates a 3D geological model
Import seismic data from ASCII	Imports seismic data from an ASCII file
Seismic calculator	Opens a tool for performing calculations on seismic data
Workflows	Opens the workflows tool for building and running processing sequences
IMPORT SEISMIC	
Add dataset	Adds a new seismic dataset to the project without loading it
Add and load dataset	Adds and immediately loads a new seismic dataset to the project
Load dataset	Loads a previously added seismic dataset into the project
Load all datasets	Loads all available seismic datasets into the project
Clear dataset	Removes the selected dataset from the project
Clear all datasets	Removes all datasets from the project
HORIZONS	
Clear horizons	Removes all interpreted horizons from the project
Update horizon map	Refreshes the horizon map with the latest interpretations
Save horizons binary	Saves the interpreted horizons in a binary format
Load horizons binary	Loads horizons from a previously saved binary file
Import horizon ASCII	Imports horizon data from an ASCII file
Export current horizon ASCII	Exports the currently selected horizon to an ASCII file
Import horizon (charisma format)	Imports horizon data in Charisma format
Export current horizon (charisma format)	Exports the currently selected horizon to Charisma format
Load external topography	Loads an external topography file for use in interpretation
Open map calculator	Opens a tool to perform calculations on maps
Clear calculated maps	Removes all maps that were generated through calculations from the project
Import checkpoints ASCII	Imports checkpoints from an ASCII file
Export checkpoints ASCII	Exports checkpoints to an ASCII file
Create unconformity map	Generates an unconformities map
WELLS	
Clear wells	Removes all well data from the project
Import wells headers from text file	Imports well headers from a text file
Export wells headers to text file	Exports well headers to a text file
Import wells inclinometry	Imports well inclination data
Export wells inclinometry	Exports well inclination data

© 2026 Geomage 2003 LTD

Import wells trajectory	Imports well trajectory data
Clear wells inclinometry	Removes all well inclination data from the project
Import wells markers	Imports well marker data
Export wells markers to text file	Exports well markers to a text file
Export well-tied horizons-markers to text file	Exports the well markers and horizons for tied wells to a text file
Clear wells markers	Removes all well marker data from the project
Load LAS files	Loads well log data from LAS files
Clear LAS files	Removes all loaded LAS files from the project
Load check shots	Loads check shot data for wells
Clear check shots	Removes all check shot data from the project
Export all check shots	Exports all check shot data to a file
Export wells comments to text file	Exports well comments to a text file
Import wells comments from text file	Imports well comments from a text file
Clear all wells comments	Removes all well comments from the project
Wavelet analysis	Performs wavelet analysis on well data
Log calculator	Opens a tool for performing calculations on well logs
Save synthetic trace to LAS	Saves synthetic seismic traces to an LAS file
Log Loading Wizard	Opens a wizard for loading well log data
Import WITSML files	Imports well data from WITSML files
Export LAS files	Exports well log data to LAS files
Markers attributes calculator	Opens a tool for calculating attributes on well markers
Grid calculator	Opens a tool for calculating a new property of the current 3D grid from its existing properties
Manage well templates	Manages well display and data templates
PETROPHYSICS	
Base properties	Calculates base petrophysical properties
Shale Volume	Calculates the shale volume (Vshale)
Porosity	Calculates porosity
Saturation	Calculates water saturation
Mineral Volume Estimation	Estimates mineral volumes
Pore pressure	Calculates pore pressure
Lithology interpretation	Performs lithology interpretation
Edit LAS curve	Edits LAS curve data
ML Log prediction	Predicts well logs using machine learning
VELOCITY MODEL	
Create depth velocity	Generates a depth velocity model using well markers and time horizons

© 2026 Geomage 2003 LTD

Import depth velocity model	Imports a depth velocity model from a file
Update depth velocity by well tie	Updates the depth velocity model using well data
Clear all well-tied maps	Removes all maps that are tied to well data from the project
Pine creation	Creates a pine tree - tool for the accuracy estimation of the velocity model
FAULTS	
AI Faults Detection	Detects faults automatically using AI (see AI faults wizard 642)
AI Horizons Stack	Automatically stacks and picks horizons using AI
Fault modeling	Builds a fault model from interpreted faults
Structural modeling	Builds a structural model
Mesh optimization	Optimizes the model mesh
Clear faults	Removes all faults from the project
Save faults binary	Saves fault data in a binary format
Load faults binary	Loads fault data from a binary file
Import fault (charisma format)	Imports fault data in Charisma format
Export current fault (charisma format)	Exports the currently selected fault to Charisma format
HORIZON PICKING	
Run QC for selected horizon picking	Performs quality control checks on the selected horizon picking
Update 3D view	Refreshes the 3D view to reflect the latest data and interpretations
Load picture files	Loads image files into the project
Clear pictures	Removes all loaded image files from the project
Load shape files	Loads shape files into the project
Clear shape files	Removes all loaded shape files from the project
Calculate gross rock volume	Computes the gross rock volume from the seismic data
Create geo-body from gross rock volume	Generates a geological body from the calculated gross rock volume
Calculate attributes	Computes various seismic attributes for analysis
Spectral decomposition	Opens the Spectral decomposition wizard for running spectral decomposition on a seismic volume
Import map from ASCII	Imports a map from an ASCII file
Update graphs	Refreshes the cross plot with the latest data
Show object properties	Displays the properties of the selected object
Calculate reserves	Computes effective reservoir volume, oil, gas or CO2 volumes
MAP INTERPOLATION	
Settings	Opens the settings for map interpolation
Map building	Opens the Map building dialog for building a map from a horizon, well markers or a point set

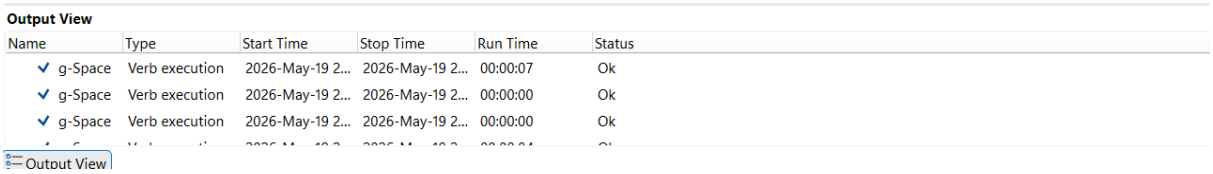
© 2026 Geomage 2003 LTD

MIS TIE SEISMIC	
Set seismic groups priority	Sets the priority for different seismic groups
Run for current Intersection	Runs mis-tie corrections for the current intersection
Run for all Intersections	Runs mis-tie corrections for all intersections
Clear shifts or rotations	Removes all applied shifts or rotations from the seismic data
Import cross points table from ASCII	Imports a cross points table from an ASCII file
Export cross points table to ASCII	Exports the cross points table to an ASCII file
Auto fit volumes	Automatically fits the seismic volumes together
Calculate volumes intersections	Computes the intersections of seismic volumes
Update lines intersections	Updates the intersections between seismic lines
PRE-STACK	
Run AVO	Runs Amplitude Versus Offset (AVO) analysis on pre-stack data
Calculate AVO maps	Generates AVO attribute maps from pre-stack data
Calculate AVO seismic attributes	Computes various AVO seismic attributes for analysis
Clear mute	Removes all mute information from the pre-stack data
Load mute	Loads mute information into the pre-stack data
Save mute	Saves mute information for the pre-stack data
DATABASE	
Set DB connection	Establishes a connection to the database
Clear references	Removes all references to database entries from the project
Sync settings	Opens synchronization settings with the database
Export data to DB	Sends the current project data to the database
Import data from DB	Retrieves data from the database into the project
Delete DB data	Deletes selected data from the database
Set gPlatform connection	Establishes a connection to g-Platform
Import data from g-Platform	Imports data from g-Platform into the project
Export data to g-Platform	Exports project data to g-Platform

3.2.8 Output View

The **Output View** is a dockable panel that concentrates every status message g-Space produces while running a procedure or background task. It is the primary place to watch long-running operations, confirm that an import finished correctly, and read the description of any error or warning raised by the engine.

© 2026 Geomage 2003 LTD



What appears in the panel

- **Name / Type / Start Time / Stop Time / Run Time / Status** — each row is one executed task; Type identifies the kind of action (Verb execution, View action, Data manager action, etc.), Status shows Ok/error at a glance.
- Double-clicking a row opens a details dialog with the full message log for that task, filterable by severity: **Debug**, **Log**, **Benchmark** (timing/throughput), **Warning**, **Error** and **Critical**.
- Right-click the list and choose **Filter by type...** to show only tasks of the selected types.

Typical uses

- Following the progress of a velocity-model build, seismic-attribute calculation or volumetric computation.
- Diagnosing data-loading problems — if a project shows inconsistent horizons, seismic, or wells after an import, the Output View typically carries a descriptive message. See also Service Tasks 526.
- Auditing database operations — connect / sync / import / export messages are all written here.

Showing and hiding the panel

Like the other dock panels, the Output View can be toggled from the main window's view menu and docked to any edge of the application. When closed, status messages still accumulate and become visible the next time it is opened.

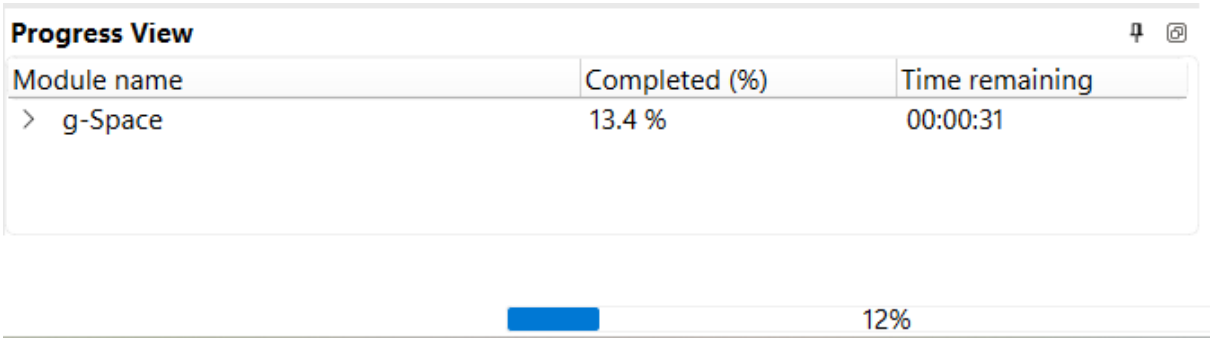
See Also

- Service Tasks 526
- Panels 123
- Data manager 124

© 2026 Geomage 2003 LTD

3.2.9 Progress View

The **Progress View** panel monitors the calculations and processes currently running in g-Space. For each process it shows the module name, the completion percentage, and the estimated remaining time. A single aggregate progress bar for all running tasks is shown at the bottom of the panel.
Long-running operations can be cancelled directly from the panel.



Multi-stage procedures can appear as nested parent/child rows, so the sub-steps of a task are shown under their parent.
For distributed or chunked computations, a **Chunk count [Completed/All]** column reports how many chunks have finished.

3.2.10 Object properties

Object properties provides general information about the selected in the Data Manager object A **Copy** button in the panel's title bar copies the currently visible table (Properties or History) to the clipboard as tab-separated text.

© 2026 Geomage 2003 LTD

Object properties

Property	Value
Label	data_3D_teapot_ft
File name	data_3D_teapot_ft.sgy
Domain	Time
Data type	Amplitude
Option	Stack
Group	PreSTM
Data dimension	3D
Bin size X	110.05
Bin size Y	110.05
Inline step	1
Xline step	1
Distance inline	37967.37
Distance crossline	20689.67
Number of CMP	64860
Inline azimuth	358.45

Other object types (maps, 3D grids, faults, polygons, Multi-Z surfaces, structural models) show their own property sets in the same panel; a <No data> placeholder appears when nothing is selected or the object has no listed properties.

List of properties for seismic data:

Label
File name
Domain
Data type

© 2026 Geomage 2003 LTD

Option — Stack or Prestack
Group
Data dimension
Bin size X
Bin size Y
Inline step
Xline step
Distance inline
Distance crossline
Number of CMP
Inline azimuth

Bin size X and **Bin size Y** show spatial distances between neighboring seismic bins in the project's distance units. Bin size Y, Inline step and Xline step are shown only for 3D seismic (2D lines show Bin size X only). Minimum offset and Maximum offset appear for prestack gathers; From azimuth and To azimuth appear for 2D prestack gathers.

List of properties for faults:

Name
User
Created — the date and time the fault was created, as YYYY-MM-DD HH:MM:SS
Source AI volume — the seismic volume used for AI fault detection
Geological type — Unknown, Normal, Reverse, Strike-slip, Strike-slip + Normal offset or Strike-slip + Reverse offset
Sticks — the number of sticks in the fault

This set appears when a fault is selected in the Data Manager. **Created** shows an em dash when the creation time is unknown, and **Source AI volume** shows an em dash for faults that were not produced by AI fault detection. The **Free sticks** section and individual sticks show the <No data> placeholder.

List of properties for maps:

Name
User
Step X — the distance between neighboring map nodes along the X axis, in the project's distance units
Step Y — the distance between neighboring map nodes along the Y axis, in the project's distance units
Number of cells — the total number of nodes in the map grid
Number of cells with values — the number of those nodes that hold a valid value
Minimum value

© 2026 Geomage 2003 LTD

Maximum value
Delta — the maximum value minus the minimum value
Mean — the average of the valid values

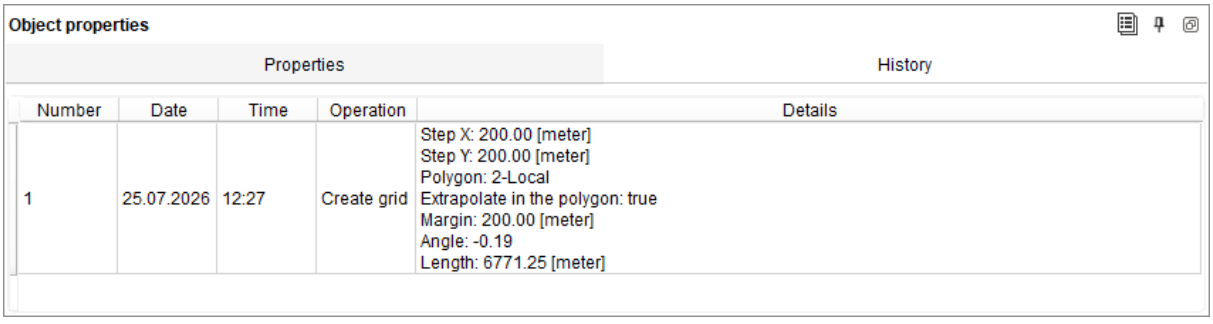
This set appears when a map is selected in the Data Manager. Unlike wells, seismic data and polygons, maps have no **Properties** command in the Data Manager context menu: the panel is filled as soon as the map is selected. **User** shows an em dash when the map has no user.

Minimum value, **Maximum value**, **Delta** and **Mean** are given in the map's own units: depth units for depth, erosion, thickness, isochore and isopach maps, velocity units for velocity maps and time units for time maps; maps of other types show their values as stored. These four rows show an em dash when the map holds no valid value, and only **Name** and **User** are listed when the map data cannot be read.

For 3D Grids, the **Object Properties** panel includes an additional **History** tab. This tab lists a timestamped history of grid-affecting operations: grid creation (step size, polygon, extrapolation flag, margin, angle, length), zone edits, renames, log/property upscaling, and grid-to-grid or seismic-volume upscaling.

Each operation is shown as one table row with the columns **Number**, **Date** (DD.MM.YYYY), **Time** (HH:MM), **Operation** (a short description of the action) and **Details** (the changed parameters). The **Copy** button in the panel's title bar copies the whole History table to the clipboard.

Distance values in the **Details** column are shown in the project's current measurement units, with the unit in square brackets after the value (for example, Step X: 200.00 [meter] in a metric project). Angle is given in degrees and has no unit.



3.3 Workspace

Workspace is a graphical setup of views that used to organize working process according to user's habits. User can add unlimited number of workspaces with unlimited number of Views 233 in each Workspace.

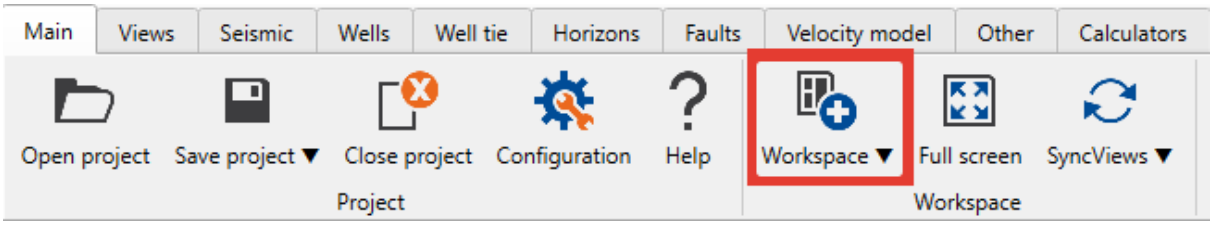
© 2026 Geomage 2003 LTD

There are two predefined Workspaces where additional Views cannot be added:
- Seismic data workspace 228 - used to load and manage seismic data loaded into the project
- Wells data workspace 232 - used to load and manage well data loaded into the project

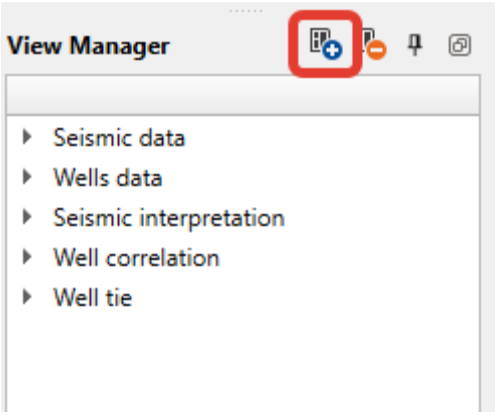
User still can arrange Views in those predefined Workspaces.

Creating new Workspace:
There are three options to add Workspace:

Option 1:
Go to Main bar 35 and click on Workspace button

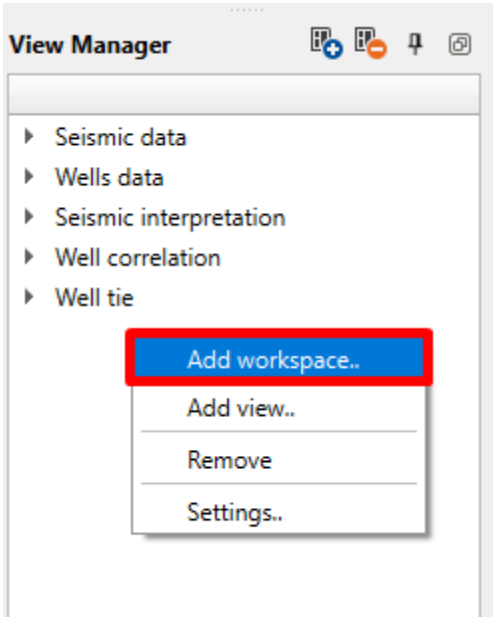


Option 2:
On the upper part of View Manager 189 click on the button add Workspace



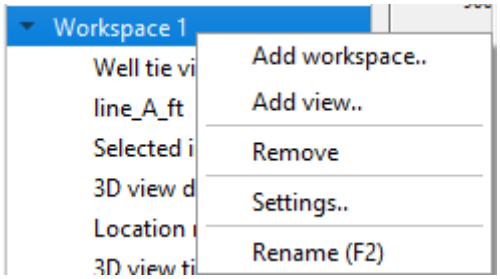
Option 3:
Right click mouse button on the View Manager and choose add Workspace

© 2026 Geomage 2003 LTD



Operations with workspace:

By clicking right button mouse on the workspace you may choose different operations you want to do with it:

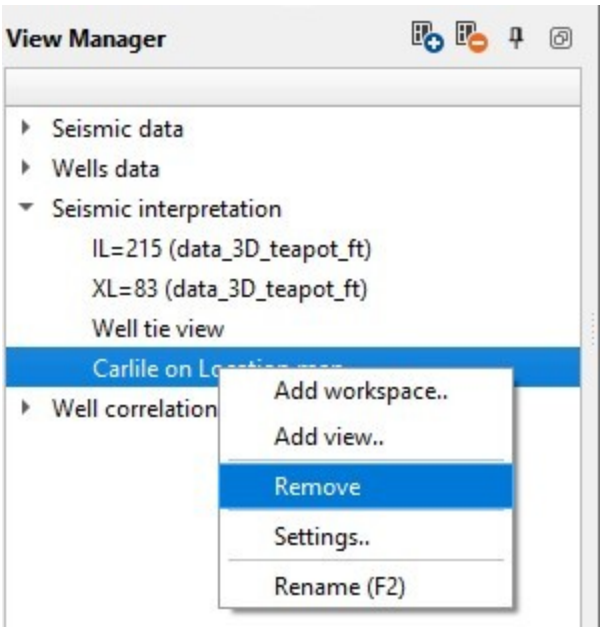


You can perform the following operations:

- Add workspace (another one)
- Add view in current workspace
- Remove workspace
- Settings - Lunch Visual Settings 208 for active view in current workspace
- Rename workspace

To delete view from the workspace press right mouse button on this view and select delete:

© 2026 Geomage 2003 LTD



Working with Views

- **Open a view** – click the view type button in the Views bar 38.
- **Switch between views** – click the view tab at the top of the workspace area.
- **Close a view** – click the X on the view tab.
- **Arrange views** – drag tabs to reorder them, or tile views side by side. Tiling is possible only when the view width is not constrained to the workspace (see below).
- **Full screen** – press F11 to toggle full-screen mode for the active view.

View width in the workspace

In g-Space the **Constrain view width to workspace** option is enabled by default: the active view is sized to the workspace area, a view cannot be made wider than the workspace, and the workspace has no horizontal scroll bar. This default applies to a new installation of g-Space. If the option was already set on the computer, the saved choice is kept.

To change it, select **Options ...** on the **Tools** tab of the main menu and, on the **General** tab of the dialog, clear the **Constrain view width to workspace** check box in the **Views** group. Views then keep their own window frame, can be made wider than the workspace, and the workspace scrolls horizontally. The change takes effect immediately.

Displaying data in a view

1. Open or select the target view in the workspace.
2. In the Data Manager 124, check the box next to the data you want to display.
3. The data appears in the active view.
4. Use the **Object** tab in Visual Settings 206 to adjust the display of the data.
5. Use the **View** tab in Visual Settings for additional appearance customization.

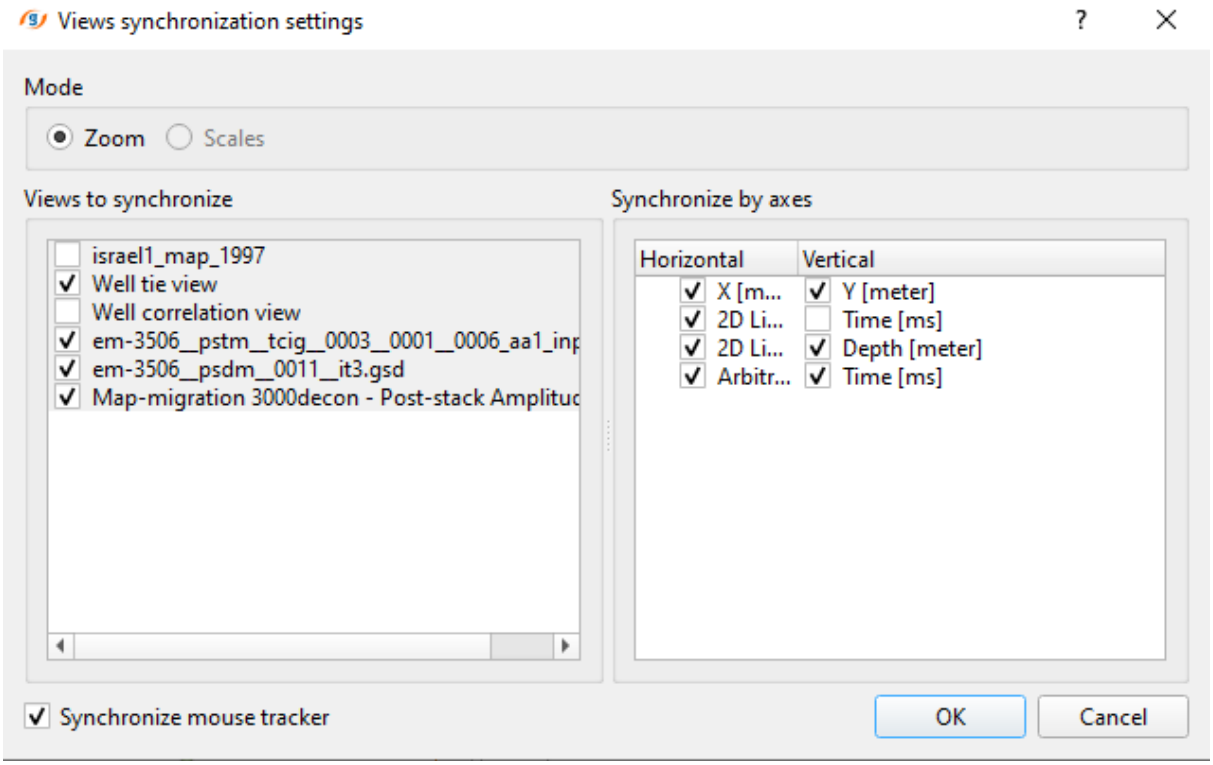
© 2026 Geomage 2003 LTD

Each view has a local toolbar at the top that gives quick access to the most useful tools for that view; the available tools change depending on the view type and the data shown.

3.3.1 Views synchronization

g-Space supports views synchronization

Go to Ribbon 33 -> Main bar 35 and on button SyncViews press down arrow to launch synchronization settings

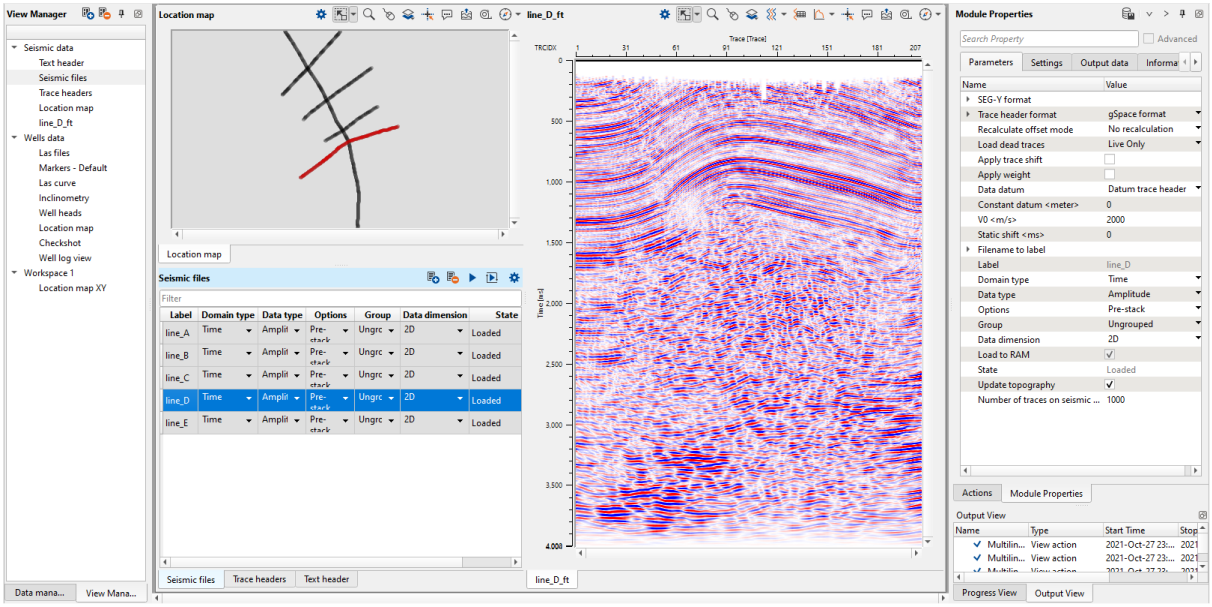


3.3.2 Seismic data workspace

Selection of Seismic data workspace in View manager activates corresponding workspace that include several views required for QC of the import process for seismic data.

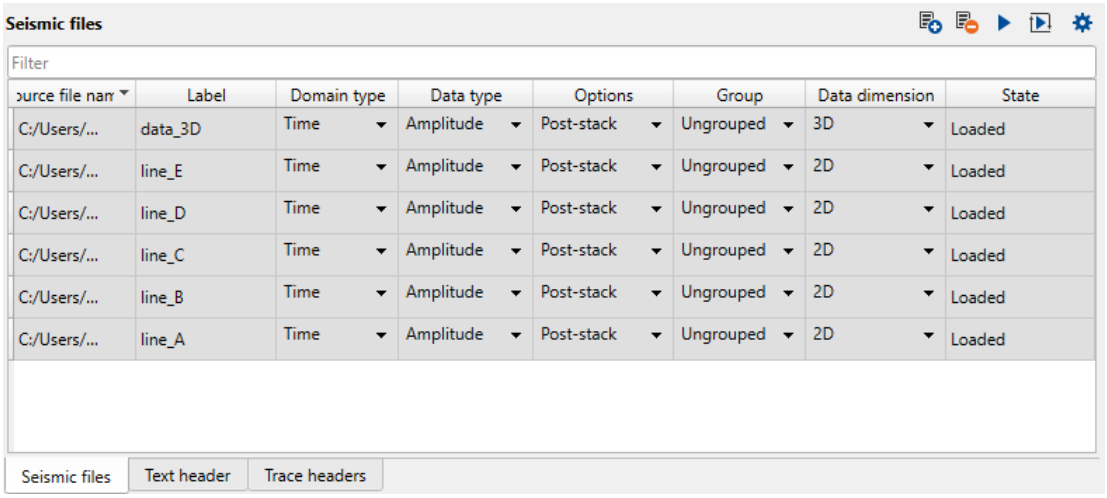
For more information about seismic data loading please refer to import seismic 373

© 2026 Geomage 2003 LTD



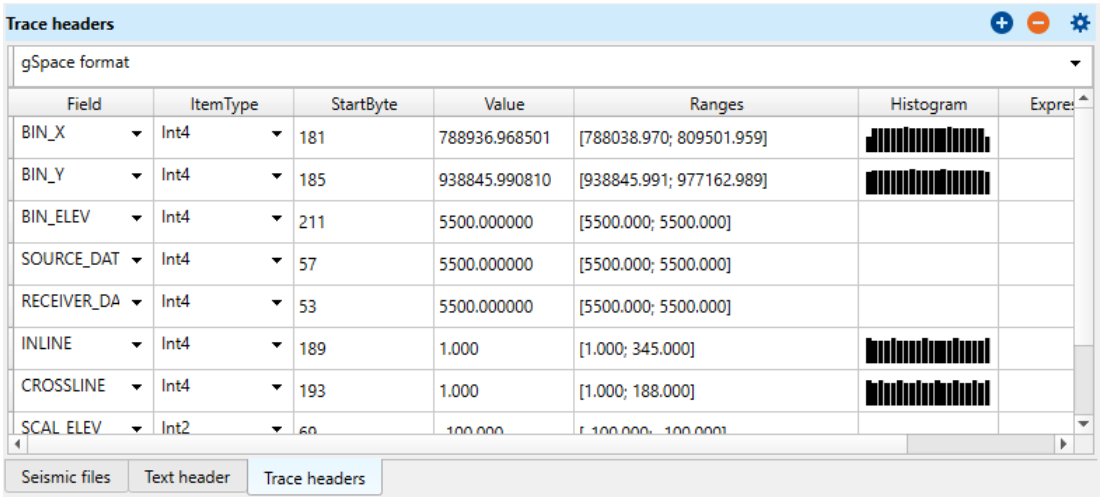
Workspace includes 2 views for settings:

- **Seismic files** view - contains all loaded sesimic files:



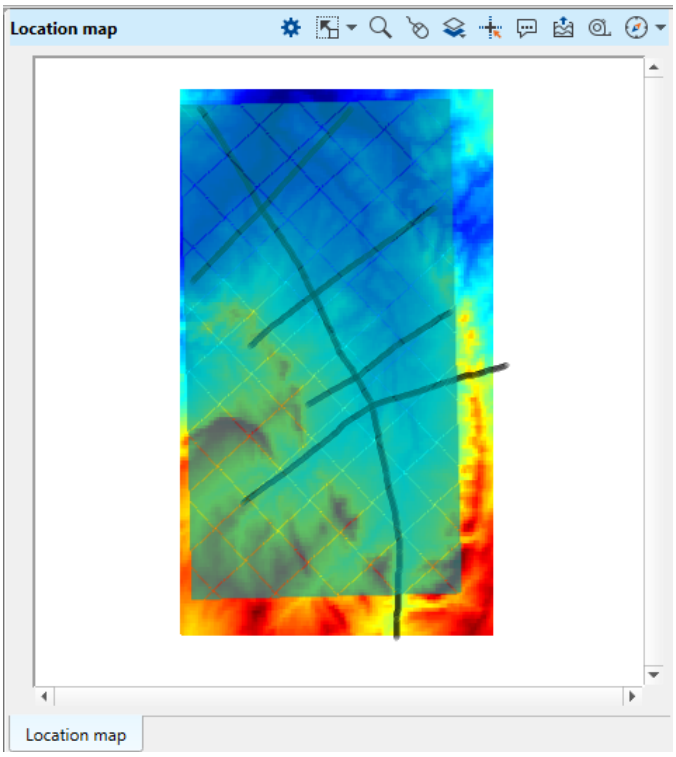
- **Trace headers** view - allows to set bins positions for loaded files:

© 2026 Geomage 2003 LTD



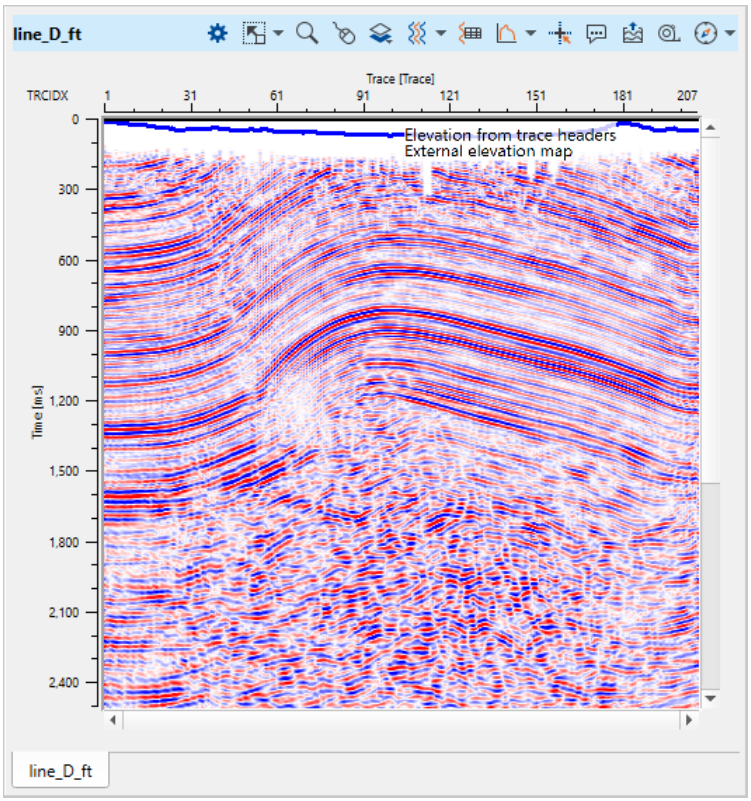
And 3 views for visualization:

- **Location map** - map with loaded seismic files and topography

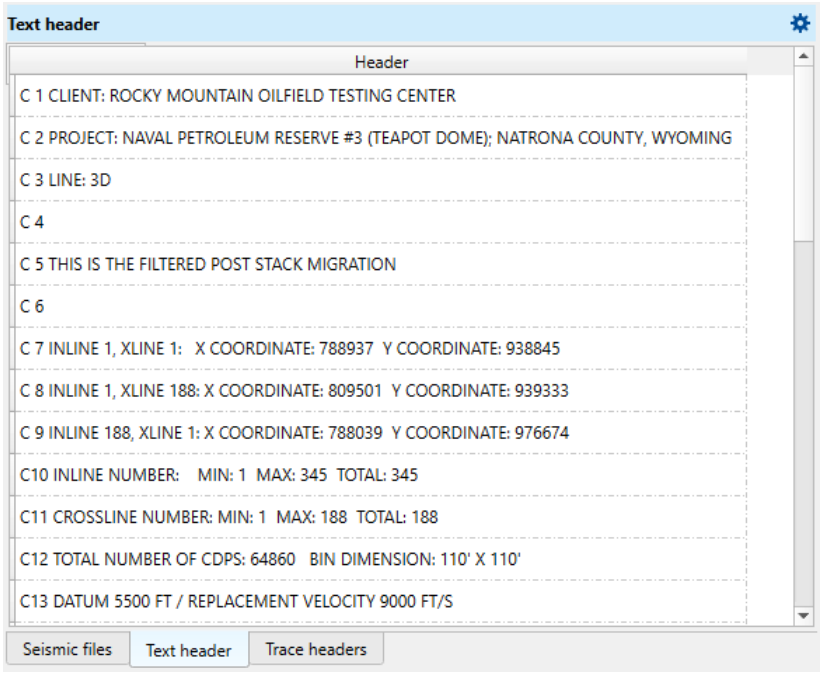


- **Section view** - traces visualization of the selected seismic file

© 2026 Geomage 2003 LTD



- **Text header** view - to view the seismic file text header

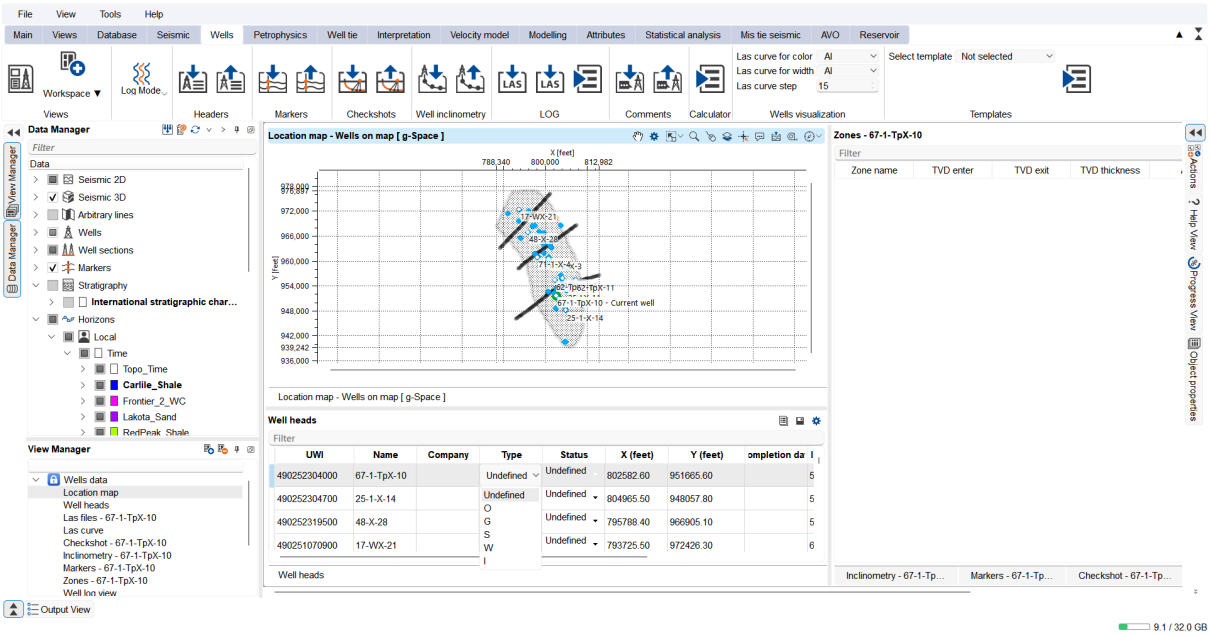


© 2026 Geomage 2003 LTD

3.3.3 Wells data workspace

Selection of Wells data workspace View manager activates corresponding workspace that include several views required for QC of the import process for all types of well data.

For more information about well data importing please refer to Import wells 391



The predefined **workspace [Wells data]** includes following views:

Location map view - shows all wells that currently loaded into the project. That view reflects:

- type and status of the well - icon of the well according (description of codes for well status can be found in https://www.ogsrlibrary.com/definitions_well_status_oil_gas_ontario)
- existence of markers loaded into the well - blue circle
- existence of checkshot information - violet circle
- existence of well logs loaded from LAS files - orange circle
- existence of deviated inclinometry of the well - black line drawn from the well location

Clicking a well on the Location map refreshes all the workspace tables for that well, allowing rapid QC.

Well heads view - shows all wells header information loaded into the project. The table columns are UWI, Name, Company, Type, Status, Icon, X, Y, Completion date, Total depth (MD),

© 2026 Geomage 2003 LTD

Kelly bushing and Ground level. To edit the classification, choose a value from the **Type** or **Status** drop-down in the table. In the **Icon** column, choose **From Type and Status** to keep the automatically derived map symbol, or choose a specific icon to use that symbol on the Location map.

Markers view - shows all markers loaded for selected (current) well. The title of the view shows the name of the current well. For each marker the table lists the attributes Marker, MD, TVD, TVDSS, TWT, X, Y, Age and Type. Any custom marker attributes defined on the marker set appear as additional columns.

Zones view - shows the zones of the project with the interval each of them covers in the selected (current) well. The title of the view shows the name of the current well. For each zone the table lists Zone name, TVD enter, TVD exit, TVD thickness, Age and Type. TVD enter and TVD exit are the depths at which the well enters and leaves the zone, TVD thickness is the interval between them, Age is taken from the top marker of the zone, and Type is always Zone. A cell stays empty when the value is not available: when the well has no pick for the top or the base marker of the zone, and for TVD thickness also when the base marker lies above the top one. Until a well is selected, the table lists every zone of every well and starts with an additional Well name column. The table is read-only; its values follow the marker tops the zones are built from. For more information about creating zones from markers please refer to Markers 150

Checkshot view - shows all checkshot values for the selected (current) well. The title of the view shows the name of current well. User can double click on any value and edit it.

LAS files view - shows all well logs loaded for selected (current) well. The title of the view shows the name of the current well.
LAS curve view - shows all values of selected well log of selected (current) well. User can double click on any value and edit it.
Well log view - for visualization logs and markers.

Inclinometry view - shows all inclinometry points for selected (current) well. The title of the view shows the name of current well. For each survey point the table lists MD, Inclination and Azimuth, together with the g-Space-calculated X, Y, TVD and TVDSS.

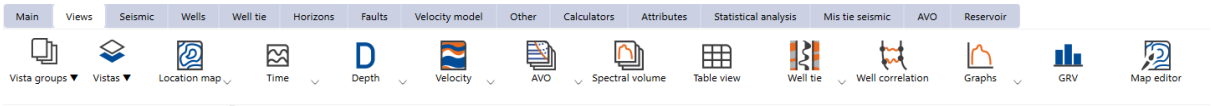
Table values can be copied with the Copy tool and pasted into Excel. Editable fields (for example checkshot values) are changed by double-clicking the cell.

3.4 Views

Views in g-Space are designed to help users visualize, quality control (QC), and manipulate various types of geological and geophysical data. Each view type offers a unique way to interact with your data, whether it's seismic, well logs, maps, or other geological models.

© 2026 Geomage 2003 LTD

Objects is a list of objects that can be visualized in any view. Within **Object Groups**, you can find multiple objects available for visualization.



g-Space provides several categories of views, each tailored to specific data visualization and analysis needs:

Seismic Views

- **Time domain** – seismic views 240 in time: 2D Line, 3D Inline, 3D Crossline, 3D Z-Slice and Arbitrary Line.
- **Depth domain** – the same seismic views with depth conversion applied.
- **Gather view** – display of prestack seismic gathers. See also AVO views 272.

Well Views

- **Well Log** – single or multi-track display of well log curves.
- Well Correlation 296 – side-by-side well sections with markers.
- **3D Well** – three-dimensional well trajectory and log display.
- Well-tie 287 – visualisation and analysis of the seismic-to-well tie.

Map & Surface Views

- Location Map 235 – GIS-type map showing wells, seismic lines, polygons and other spatial data.
- Map Editor 341 – tools for refining, annotating and enhancing maps.
- 3D View 244 – full 3D visualisation of surfaces, models, wells and volumes.

Analysis Views

- Cross-Plot 330 – scatter plots for multi-parameter analysis.
- Histogram 327 – statistical distribution display.
- Table View 285 – visualisation and editing of tabular data.
- **Spectral Volume** – spectral decomposition analysis.
- Pine Chart 332 – velocity analysis charts.

Customizing View Appearance

You can adjust the appearance of each view, as well as the visualization parameters for objects within the view, using the Visual Settings 206 panel. This panel provides a range of settings to tailor the display to your specific needs.

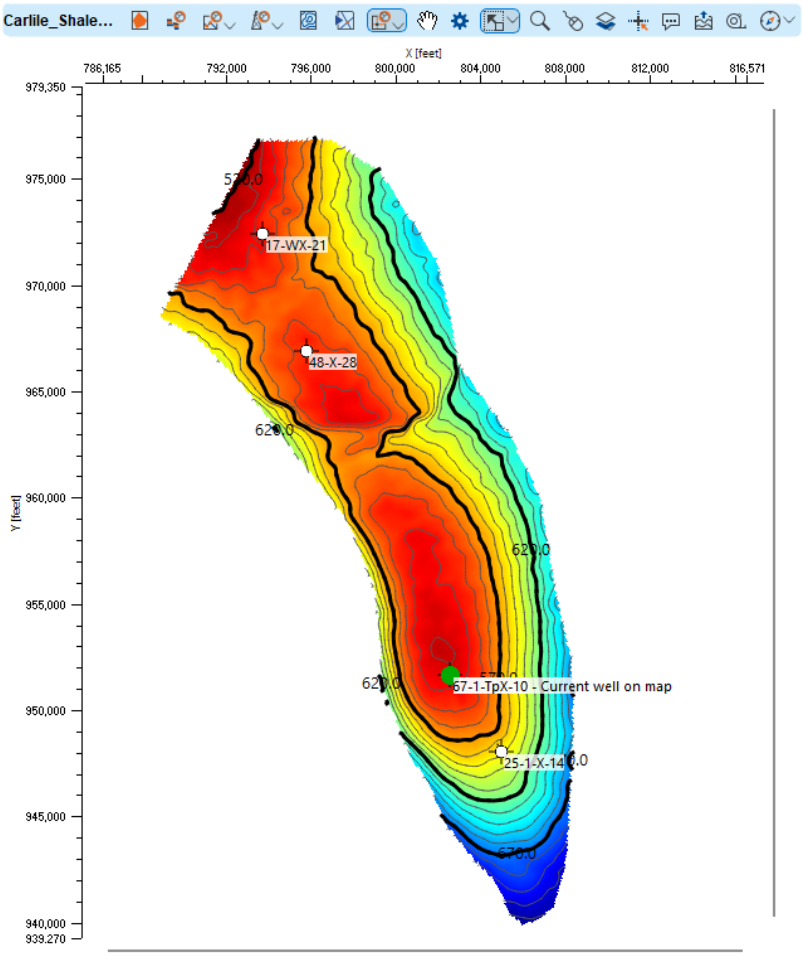
Most of the views operate similarly for zoom, scroll and selection of picking objects. For more information about mouse and keyboard controls and shortcuts refer to Controls and shortcuts 343

© 2026 Geomage 2003 LTD

3.4.1 Location map

The **Location Map** is a window that allows users to display and manage loaded GIS-type data, including seismic, well data geometry, polygons, arbitrary lines, maps, horizons, and more.

The Location Map view consists of a map canvas with a coordinate frame and a functional panel at the top of the view.



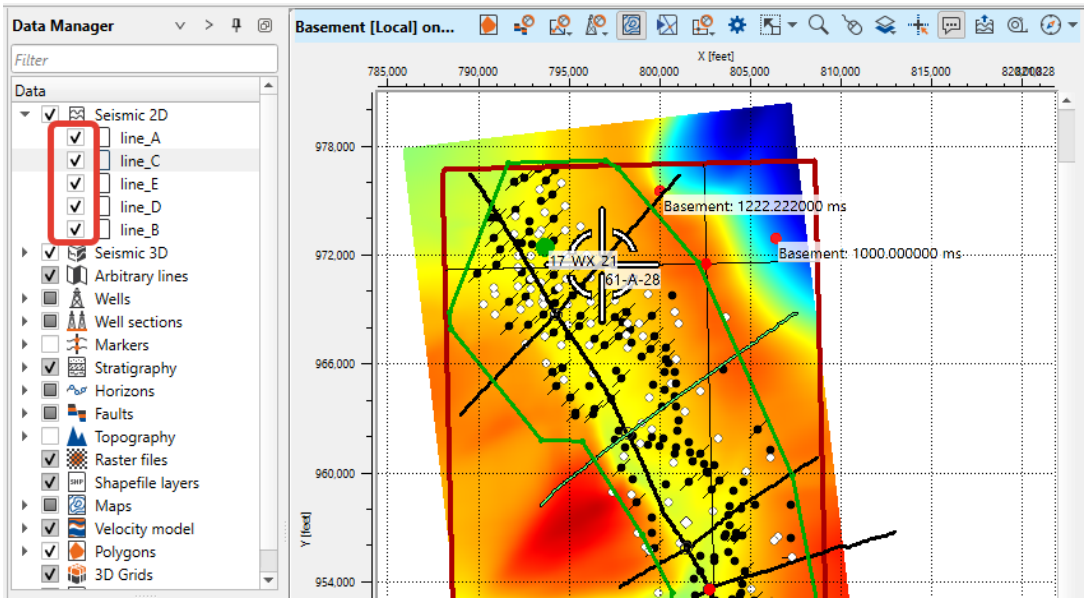
To create location map view go to the Views bar 38 in the Ribbon bar 33 and click on Location map button. A new, empty Location map view opens by default.

© 2026 Geomage 2003 LTD

The newly created view will appear in the View manager 189 in the active Workspase 224. User can place location map view by dragging it to any workspace except pre-defined once (Seismic data 228 and Well data 232).

To define the object visibility style go to the Visual Settings 206 by pressing in the view functional panel or find it in the panel 123 sets.

On the **Location map view** user can visualize any object with coordinates. To visualize the object press its checkbox in the Data manager 124.



When a fault is ticked in the Data manager, the Location map shows only the fault lines that follow the current horizon and the depth maps: **Faults on current horizon**, **Faults on depth maps**, **Fault model faults on current horizon**, **Fault model faults on depth maps**, **Structural model faults on current horizon** and **Fault contact lines**.

The other fault items are switched off by default and stay off when the fault is ticked: **All fault sticks**, **Selected fault map**, **All faults map**, **All fault model map**, **Picked fault bins** and **Fault picking**. Switch them on in **Visual Settings** on the **Object(s)** tab.

Every inline and crossline section added under the **Sections** folder of a 3D survey in the Data manager (**IL <number>** and **XL <number>**) is drawn on the Location map as a straight line spanning the survey. A newly added section appears at once; clearing its checkbox hides that line, ticking it shows the line again, and **Remove** deletes the item together with its line.

© 2026 Geomage 2003 LTD

These lines form the **All sections XY** object in the **Seismic** folder of **Visual Settings** on the **Object(s)** tab. Drag a section line on the map to move the section to the nearest inline or crossline; its item in the Data manager is renamed to the new line number. The **Selected Inline <number>** and **Selected Crossline <number>** items are not part of **All sections XY** and keep showing the currently selected inline and crossline.

Functional panel allows user to perform the following actions:

edit polygon

edit fault

edit 2D arbitrary line

edit 3D arbitrary line

current well on map

edit well section

edit checkpoints

select intersection

select 2D/3D/image bin

open Visual Settings panel

adjust view area to all visible objects

adjust horizontal scale

adjust vertical scale

set/unset zoomable

set control item

© 2026 Geomage 2003 LTD

add Object

enable mouse tracker

enable tool tip

export picture

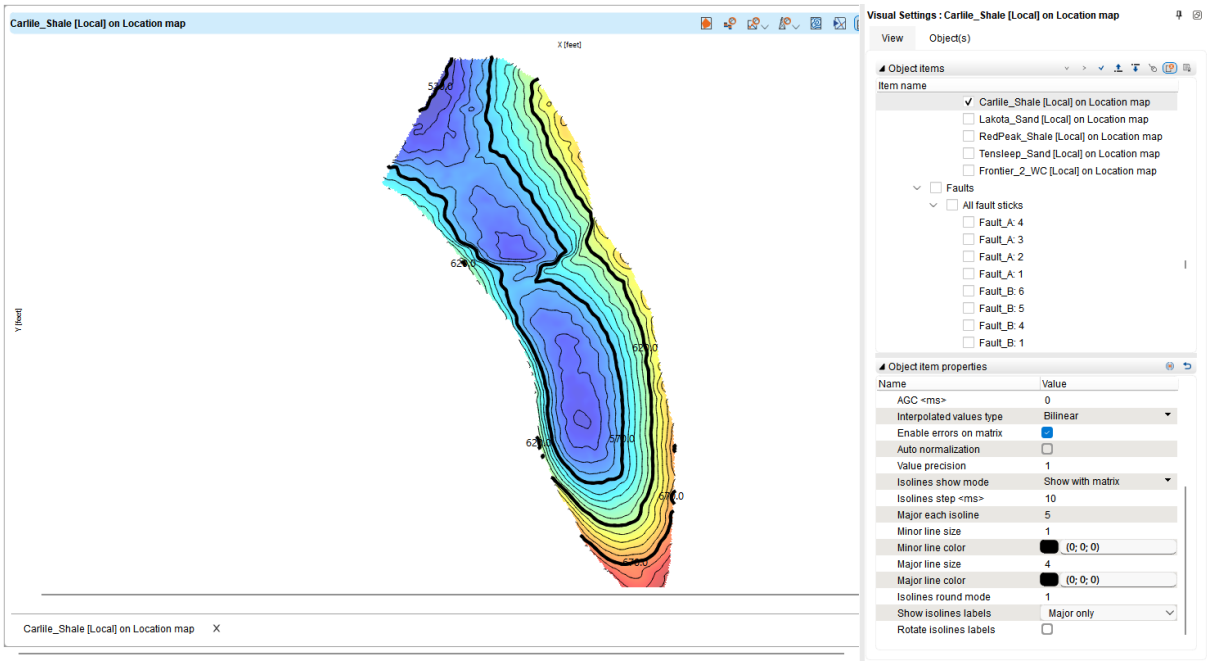
ruler tool

show compass

show minimap

show scale bar

Object Item Properties Overview



Transparency Level

- Adjusts the transparency of the displayed map.
- Range: 0 (fully opaque) to 255 (fully transparent).

© 2026 Geomage 2003 LTD

Color Bar

- Defines the color scale applied to the map.

Inverted Palette

- Reverses the color gradient of the selected color bar.

Normalization

- Defines how data values are normalized for color mapping.
- Options include *Matrix* (based on grid values).

Min/Max Range Value

- Allows users to manually set the minimum and maximum values for the color scale.

Hide Out of Range

- If enabled, values outside the defined range are hidden instead of clamped to min/max values.

Gain

- Adjusts the brightness of the map, affecting contrast between colors.

AGC (Automatic Gain Control)

- Affects amplitude scaling by normalizing the dynamic range.
- Measured in milliseconds.

Show Interpolated Values

- Enables the display of interpolated values in between original grid points.

Enable Errors on Matrix

- Highlights areas where the data matrix has inconsistencies or missing values.

Isoline Settings

Isolines show mode

- Controls the visibility of contour lines (isolines) on the map. Newly displayed maps use **Show with matrix** by default.
- **Don't show** – No isolines are displayed.
- **Show with matrix** – Isolines appear over the color-mapped grid.
- **Show isolines only** – Displays isolines without the underlying color matrix.

© 2026 Geomage 2003 LTD

Isolines step

- Defines the interval between adjacent isolines in the unit shown with the control. A smaller value increases the density of isolines. The initial step for a newly displayed map is 100 in that unit. You can change it as needed; the screenshot above shows a customized step of 10.

Major each isoline

- Defines how frequently a thicker major isoline appears (e.g., every 5th isoline).

Minor Line Size & Color

- Adjusts the thickness and color of minor isolines.

Major Line Size & Color

- Adjusts the thickness and color of major isolines.

Isolines Round Mode

- Controls rounding behavior of isoline values.
- A higher round mode reduces precision but simplifies contour labels.

Show isolines labels

- Choose **No labels**, **Major only**, or **All**.
- For label spacing, size, color, rotation, and orientation controls, see Map editing 733.

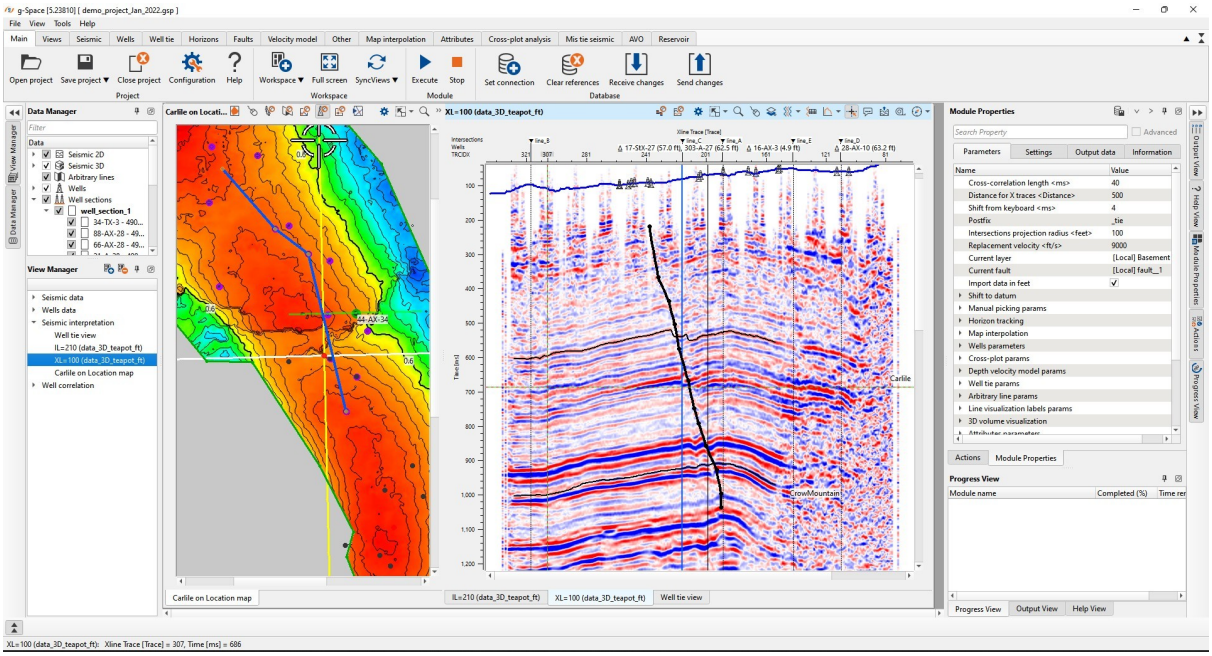
Isolines Color by Palette

- If enabled, isoline colors match the underlying color map instead of using a fixed color.

3.4.2 Seismic view

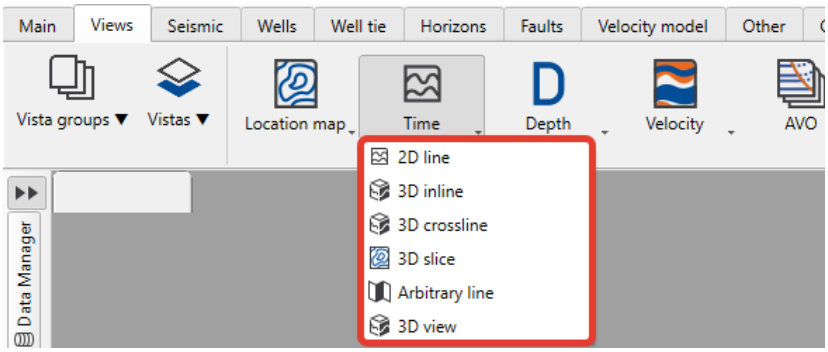
Seismic view is the set of section view windows where user can visualize seismic data and additional information such as grids, wells, markers, horizons, faults etc.

© 2026 Geomage 2003 LTD



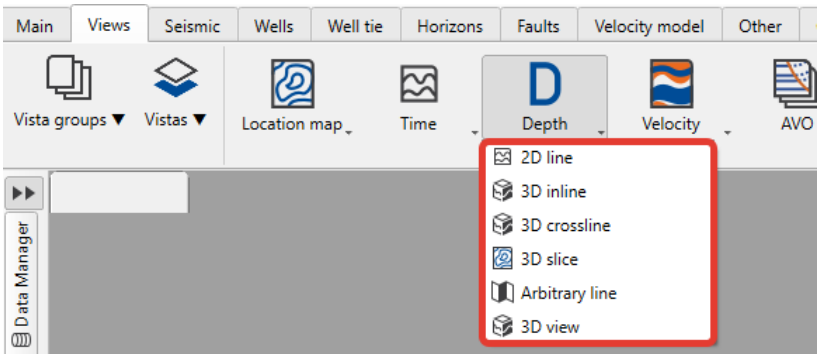
To create seismic section view go to the Views bar 38 on the Ribbon bar 33 and:

to create view in **time domain** open Time tab and choose between views: 2D line (for 2D data), 3D inline/crossline (for 3D data) or arbitrary line (for both 2D and 3D):

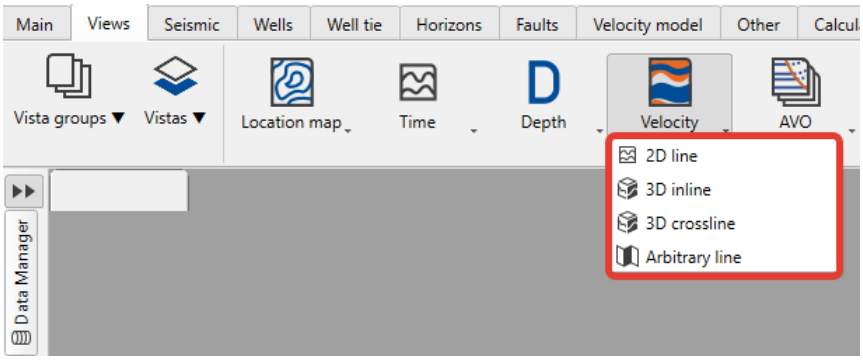


to create view in **depth domain** open Depth tab and choose between views: 2D line (for 2D data), 3D inline/crossline (for 3D data) or arbitrary line (for both 2D and 3D):

© 2026 Geomage 2003 LTD



to create view for **velocity data** open Velocity tab and choose between views: 2D line (for 2D data), 3D inline/crossline (for 3D data) or arbitrary line (for both 2D and 3D):



The newly created view will appear in the View manager 189 in the active Workspase 224. User can place Views by dragging them to any workspace except pre-defined once (Seismic data 228 and Well data 232).

To define the object visibility stile go to the Visual Settings 206 by pressing in the view functional panel or find it in the panel 123 sets.

To visualize seismic data on the seismic section view make the loaded survey active by double click on it in the Data manager.

When seismic data is loaded into a project in which no line or bin has been selected yet, g-Space selects a position at the centre of the loaded data automatically, so the section views already show seismic instead of opening empty. For 2D data a line from the middle of the loaded lines is selected and positioned on its middle trace; for 3D data the selected bin is set to

© 2026 Geomage 2003 LTD

the middle inline and crossline. Reopening a saved project restores the line or bin that was selected last instead.

Additional information about seismic visualization you can find here 587.

Functional panel allows user to perform the following actions:

faults picking

horizons picking

open Visual Settings panel

adjust view area to all visible objects

adjust horizontal scale

adjust vertical scale

set/unset zoomable

set control item

add Object

select traces

trace data

spectral analysis tool - multi selection

spectral analysis tool - single selection

enable mouse tracker

enable tool tip

© 2026 Geomage 2003 LTD

export picture

ruler tool

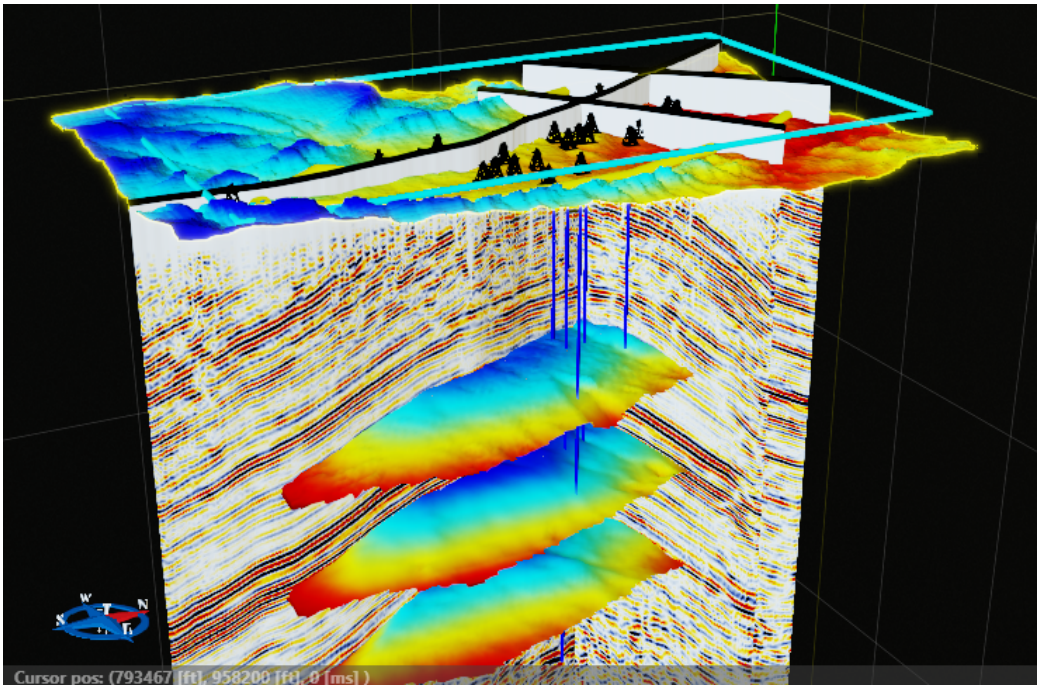
show compass

show minimap

show scale bar

3.4.3 3D view

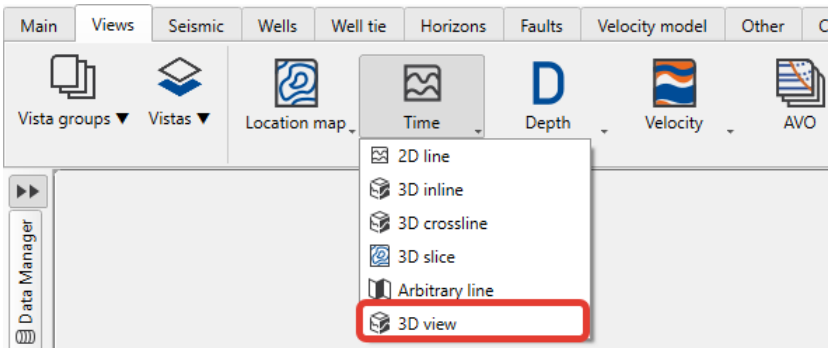
3D View is a window that allows users to operate and visualize any geophysical and geological data, such as seismic volumes, well data, polygons, grids, horizons, maps and faults, in a 3D perspective.



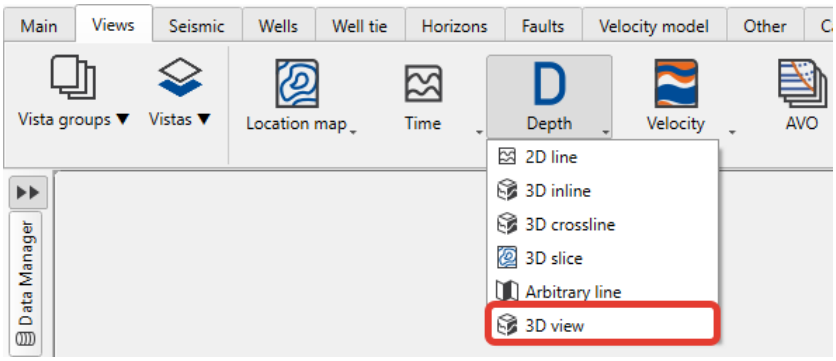
To create seismic section view go to the Views bar 38 on the Ribbon bar 33 and:

to create view in **time domain** open Time tab and select 3D view:

© 2026 Geomage 2003 LTD



to create view in **depth domain** open Depth tab and choose between views: 2D line (for 2D data), 3D inline/crossline (for 3D data) or arbitrary line (for both 2D and 3D):



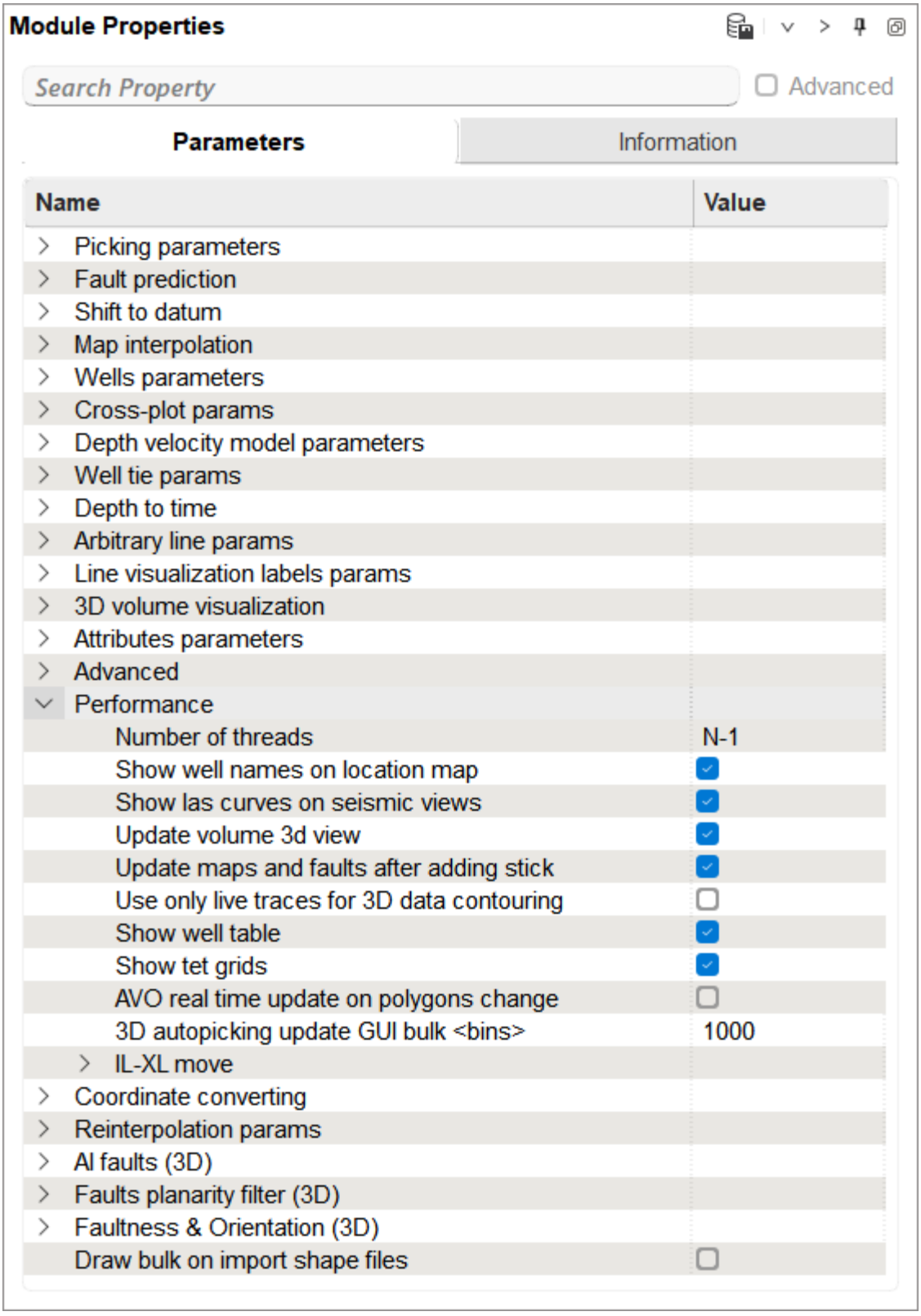
g-Space provides a clear distinction between **time and depth domains**, organizing **Horizons, Faults, and Maps** into dedicated folders for each.

For **3D visualization**, g-Space offers a feature that allows time-domain data to be displayed within the **3D depth view** using **the current velocity model**. This is especially useful during interpretation, as it enables reviewing results in time without requiring full data conversion to the depth domain.

In order to allow 3D view objects drawing, the user should adjust the **Performance** section found under Module properties 197 panel**.**

The main objective of these parameters is to optimize the performance of g-Space by unchecking them when the 3D view is not required. When more features are needed in the 3D view, the corresponding checkboxes should be selected.

© 2026 Geomage 2003 LTD



© 2026 Geomage 2003 LTD

Show well names on location map Displays all well names on the location map when checked.

Update volume 3d view Automatically refreshes displayed 3D volume content when its visibility is turned on.

Update maps and faults after adding stick Automatically recreates the grid map of horizons and updates fault sticks when new fault sticks are added.

Use only live traces for 3D data contouring By default unchecked. When checked, only live traces are used for creating contour maps.

Horizon performance

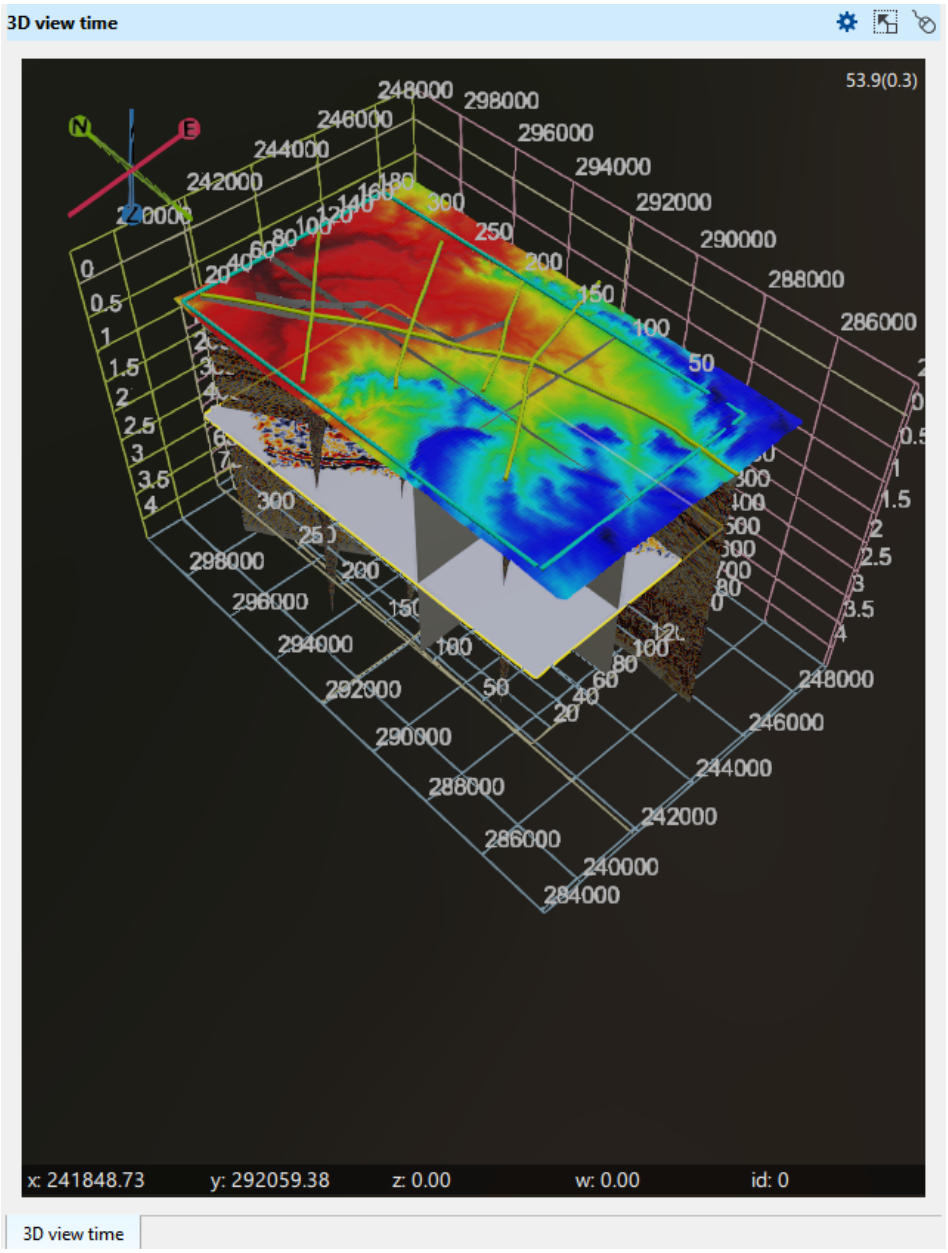
Update horizon phase maps Automatically refreshes horizon phase maps based on changes in horizon data.

Update horizons 3d view Updates the 3D horizon view whenever changes are made to existing horizons (e.g., picking, adding new horizons).

Displaying 3D volume

On the picture a few 2D lines, 3D volume, an arbitrary line, time slice, a topography map and wells are displayed on **Time 3D view**.

© 2026 Geomage 2003 LTD

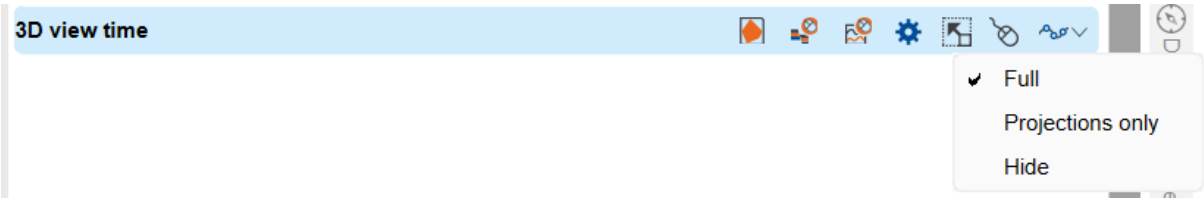


At the bottom of the display, the X, Y, and Z axes are shown, displaying the mouse position and its corresponding values. In the top right corner, the FPS (frames per second) value is displayed.

Horizon display

© 2026 Geomage 2003 LTD

The view header toolbar has a **Horizon display** button (horizon icon), which chooses how horizons are drawn in this 3D view. Its menu offers three mutually exclusive modes, and the choice is remembered for each view separately.



Full The default mode. Horizon surfaces, topographic horizons, maps and horizon attribute surfaces are all drawn.

Projections only Replaces the horizon surfaces with projection lines drawn where each horizon crosses the currently visible inline, crossline and arbitrary line sections, coloured from that horizon's own palette. The full horizon surfaces, maps, horizon attribute surfaces and topographic horizons are hidden, while horizon picks stay visible. The lines are rebuilt automatically when the visible sections are moved, added or removed, so the scene stays clean and shows only the interpretation on the sections.

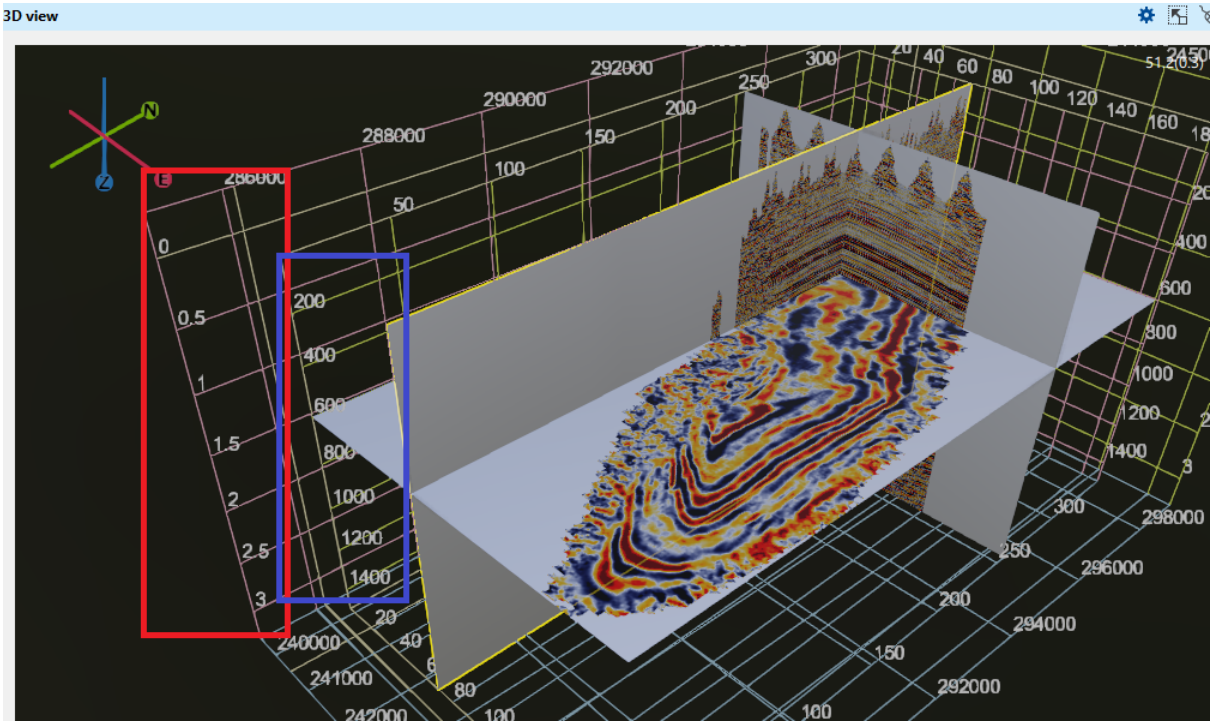
Hide Hides all horizon objects, including horizon picks.

The mode masks the Visual Settings 206 selection instead of replacing it: an object that is unchecked on the **Object(s)** tab stays hidden, and the mode only limits what the checked horizon objects may draw. Detailed per-object control therefore remains in Visual Settings 206.

Vertical scale

In 3D view, two vertical scales exist. One is inside the volume and other is outside the volume. These two vertical scales shows us two different set of values.

© 2026 Geomage 2003 LTD



The vertical scale of the inside grid (marked as blue rectangle) is showing the volume in **samples**. Like if we have 3000ms of record length and we have 4ms of sample interval, it will display like 200, 400 etc.

The vertical scale of the outside gird (marked as red rectangle) is showing the volume in **seconds**. If the record length is 3000ms then we can see it as 1s, 1.5s etc.

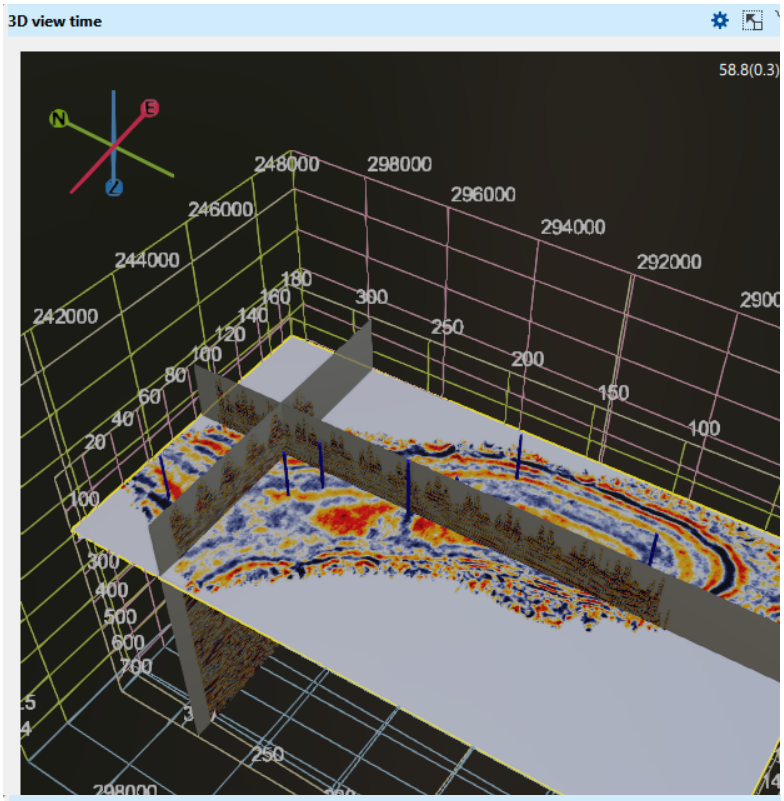
View appearance parameters and object visualization parameters can be set on the Visual Settings 206 panel as for other views.

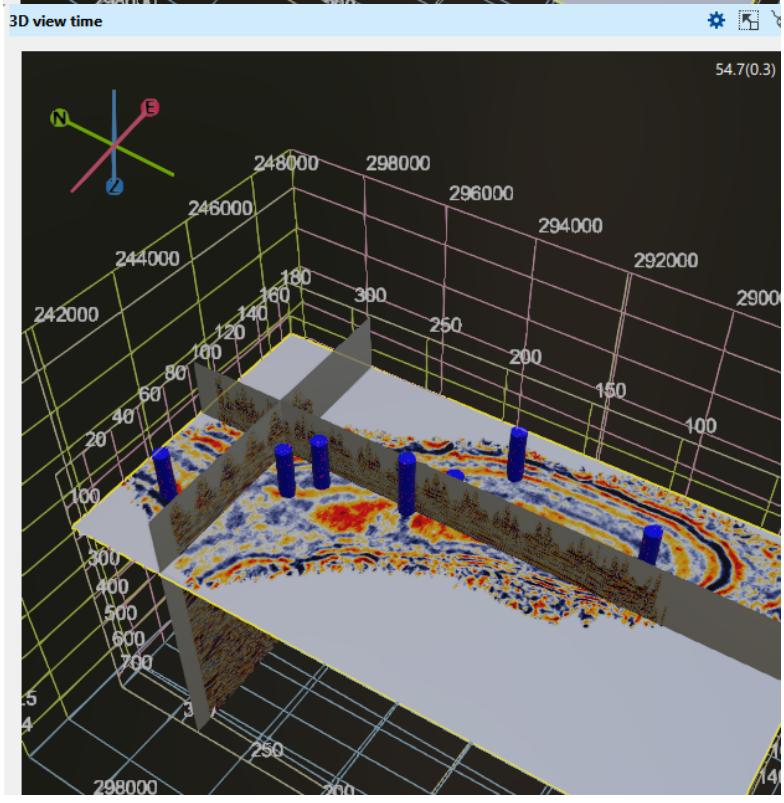
Anyway a few settings related to 3D view only described below.

View tab:

Well scale This property changes the thickness (size) of the displayed wells in the 3D view. By default it is 100. When the user changes it into 500 the thickness(size) increase.

© 2026 Geomage 2003 LTD

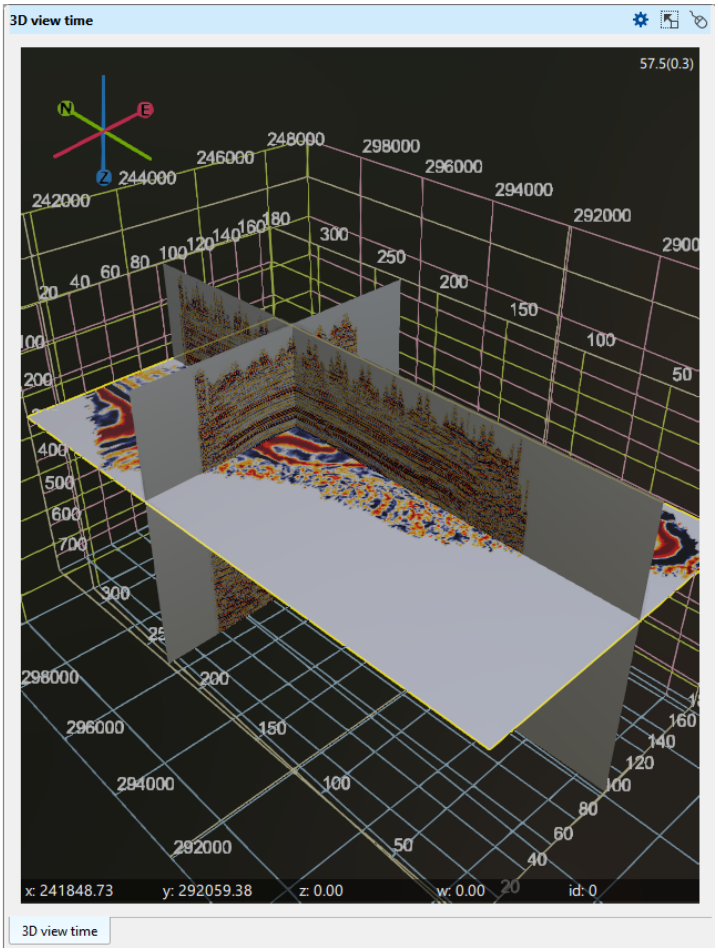




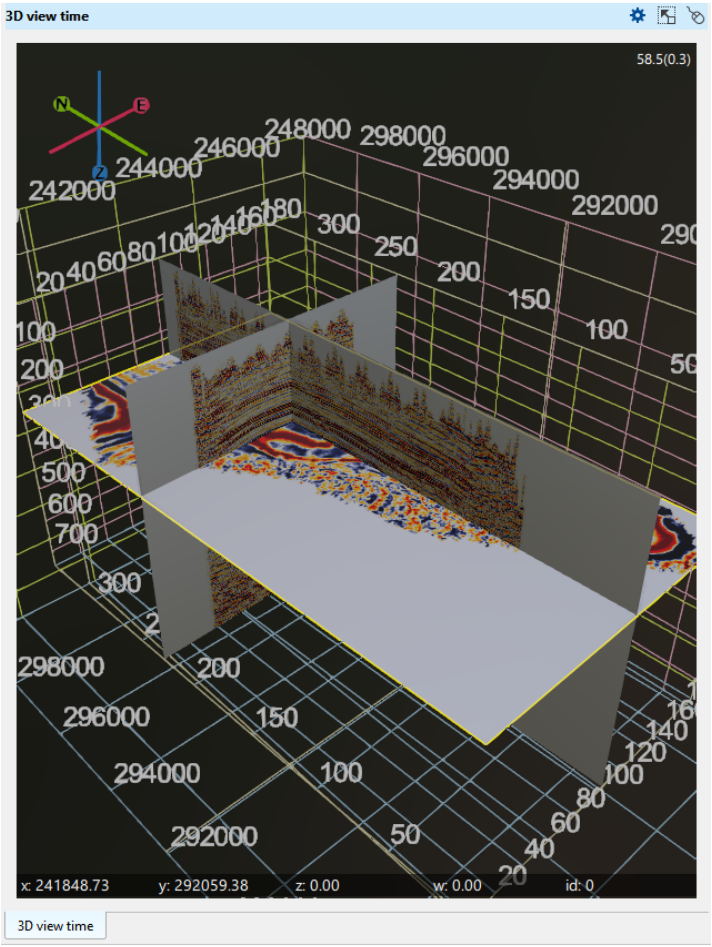
© 2026 Geomage 2003 LTD

Labels size - The user can change the size of the X, Y, Inline, Crossline, Time/Depth slice labels. By default it is 1. Here we are changing it from 1 to 1.5. We can see the changes in the below image.

© 2026 Geomage 2003 LTD

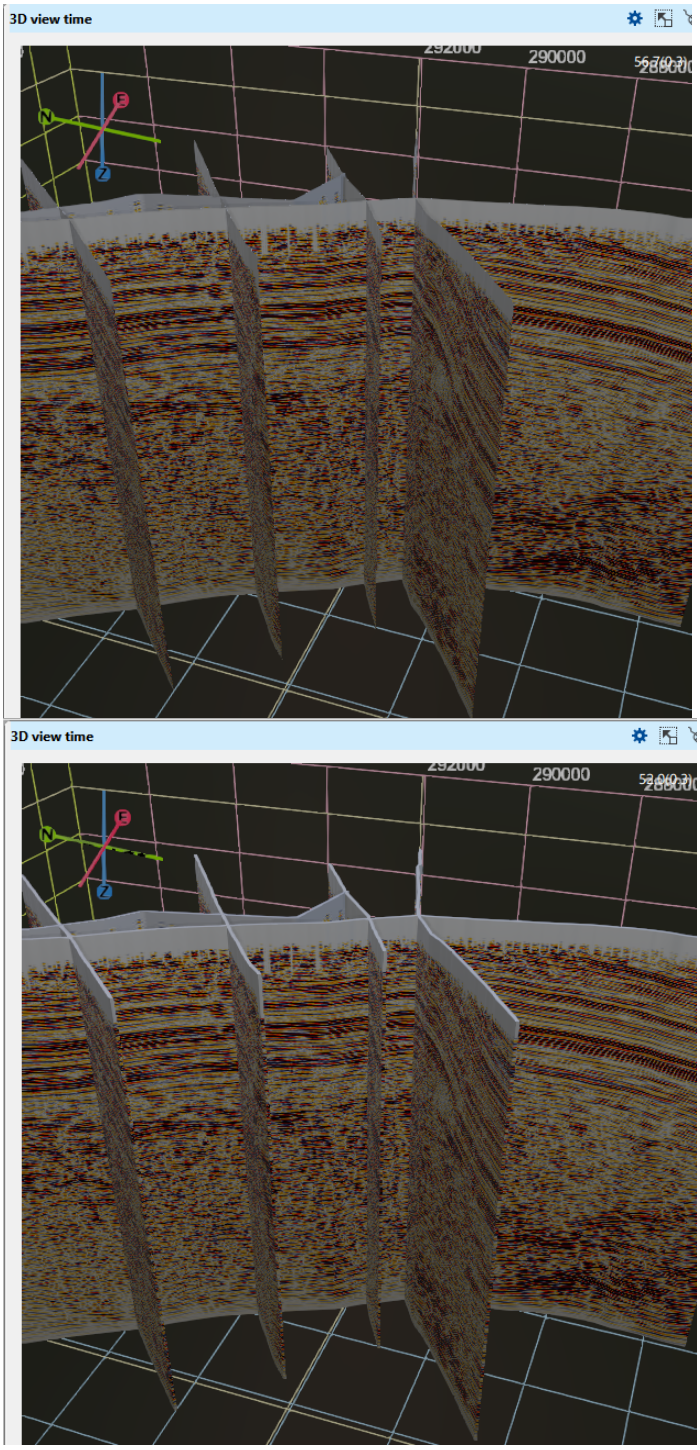


© 2026 Geomage 2003 LTD



Primitive thickness This parameter increases the display thickness of the 2D, 3D(Inline/crossline/time slice) and arbitrary line. In this instance, we have 1 as by default and changed that into 50 and we can able to see the thickness of the 2D lines as shown below.

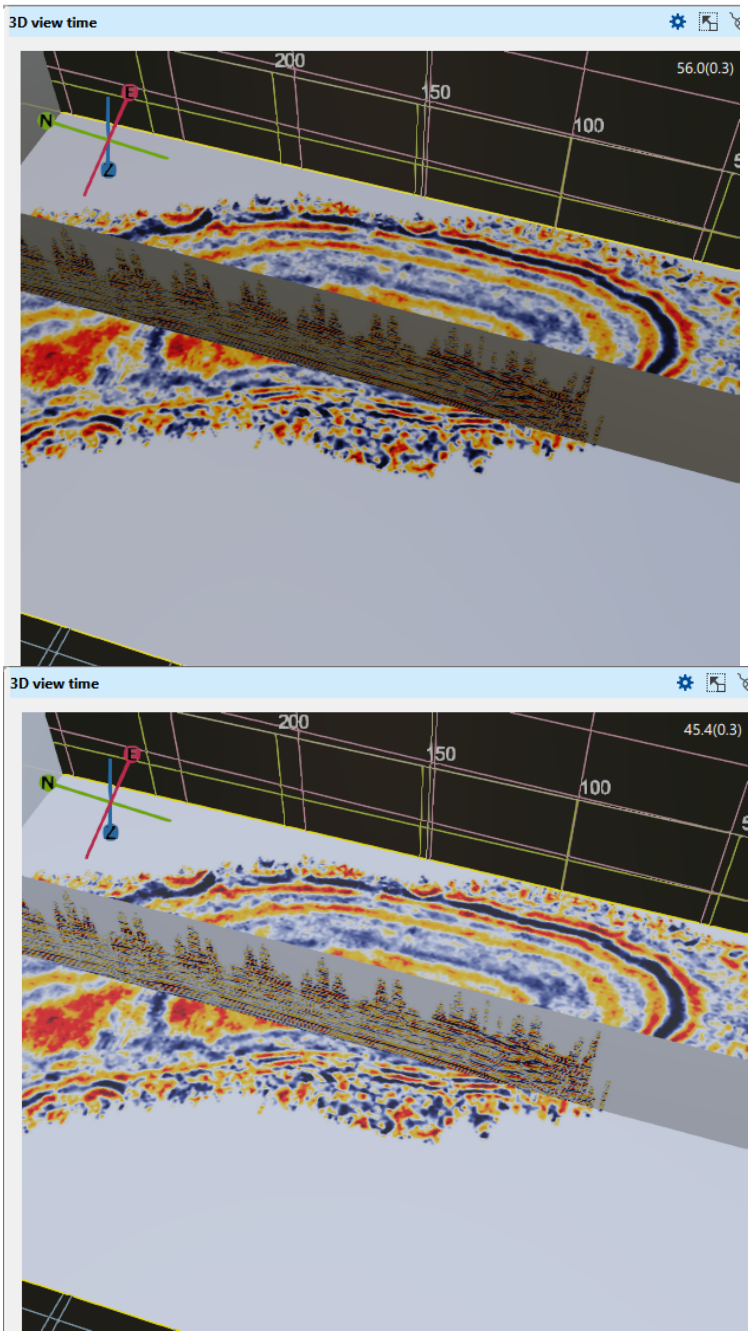
© 2026 Geomage 2003 LTD



Points size It is a scalar for all 3D points like well markers, well rigs etc.

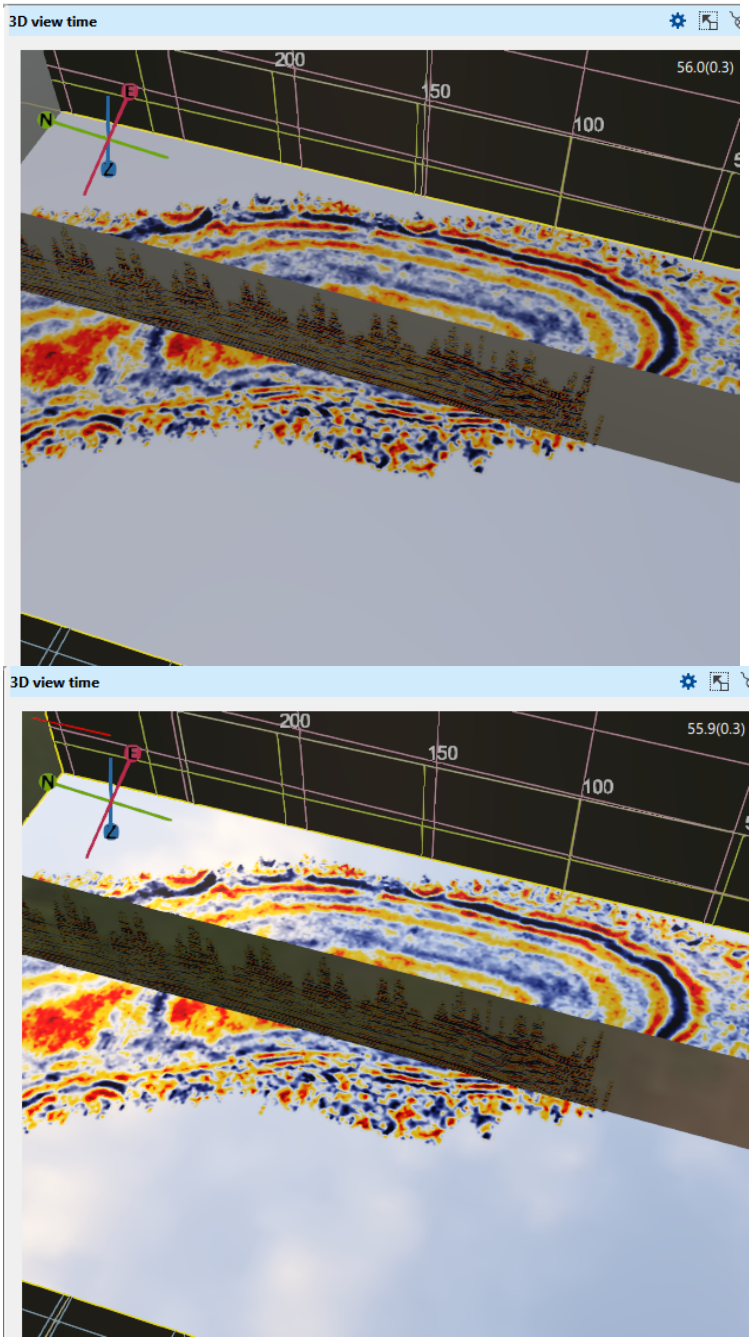
© 2026 Geomage 2003 LTD

Metalness This parameter is similar to brightness/contrast. By default it is 0. If the user changes this to a higher value, say 0.2, the contrast increases as shown below.



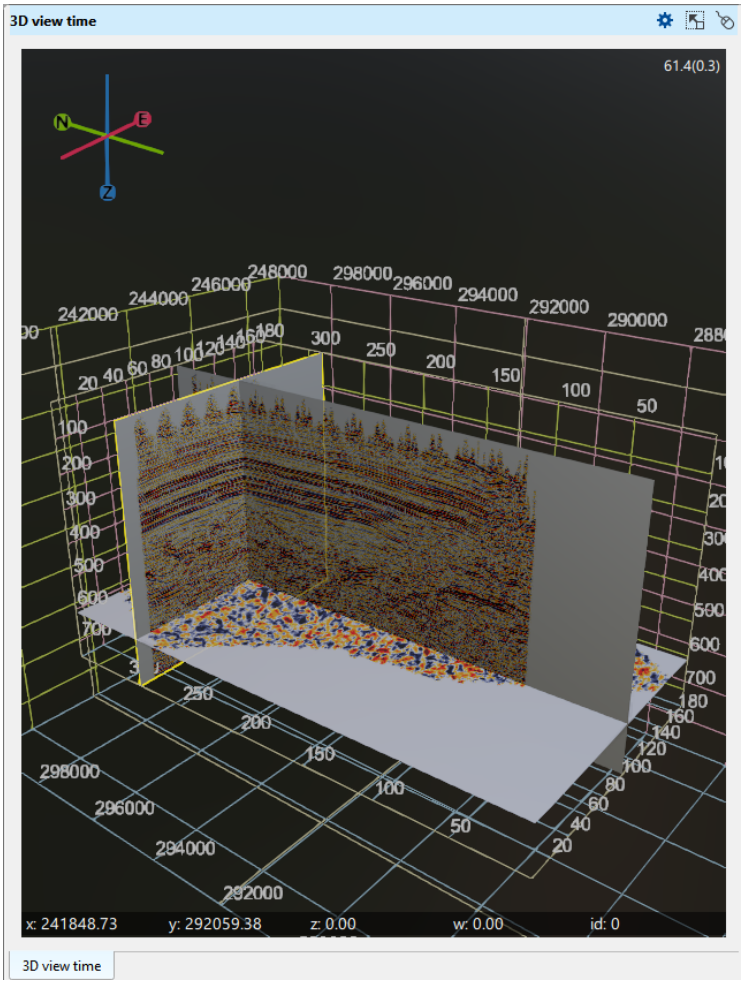
Roughness This parameter is also similar to Metalness. The default value is 0.01. It can act like brightness/contrast as shown below.

© 2026 Geomage 2003 LTD

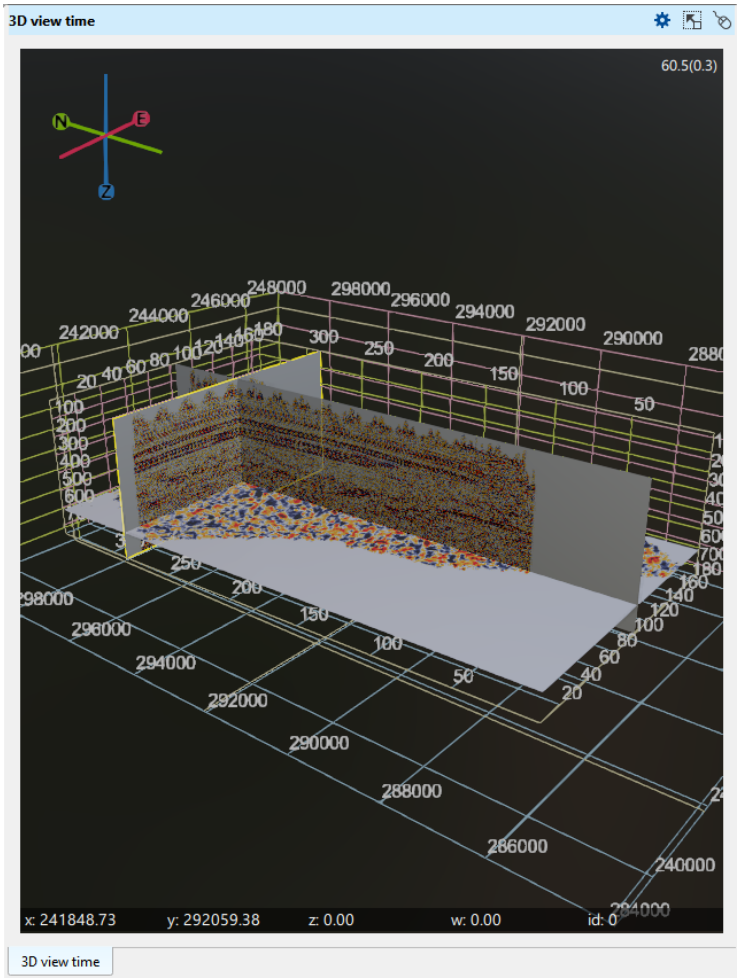


Vertical axis scale adjusts the volume's size by either expanding or compressing it along the vertical axis. The default value is 2000. In the display, the left side shows the volume with a vertical axis scale of 2000, while the right side shows the same volume with a value of 1000. In the latter case, the vertical scale appears reduced (zoomed out/in), effectively altering the vertical axis's scalar value.

© 2026 Geomage 2003 LTD

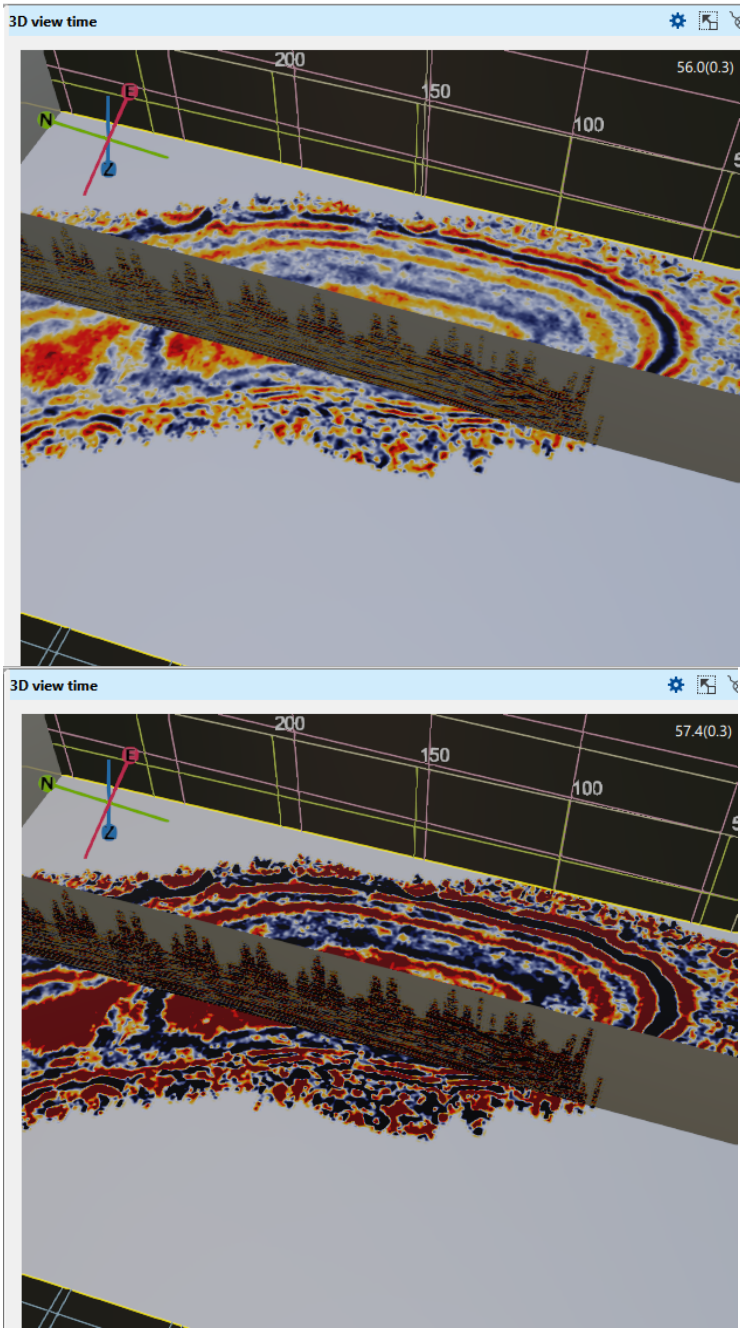


© 2026 Geomage 2003 LTD



Attribute multiplier This parameter is like adding gain to the selected display. By default it is 1. When we change it from 1 to 5, it looks like the below shown image.

© 2026 Geomage 2003 LTD



Perspective It is related to 3D projection. By default it is checked, if it is unchecked the object (in that case like a well marker) the one closer to the focus point is bigger than the one far way from the focus point.

Show global grid Shows the coordinate box drawn around the whole scene. By default it is checked.

© 2026 Geomage 2003 LTD

Show local grids Shows the grids drawn with individual objects, such as the box around the seismic volume and the grids of the well log tracks. By default it is checked. Unchecking both **Show global grid** and **Show local grids** hides all the grids in the view, and they stay hidden until you check them again.

Show grid labels Shows the value labels along the grid axes. By default it is checked.

Show handler labels Shows the labels of the objects and their handlers in the scene. By default it is checked.

Horizon display mode Repeats the mode chosen with the **Horizon display** button on the view header: 0 for **Full**, 1 for **Projections only** and 2 for **Hide**.

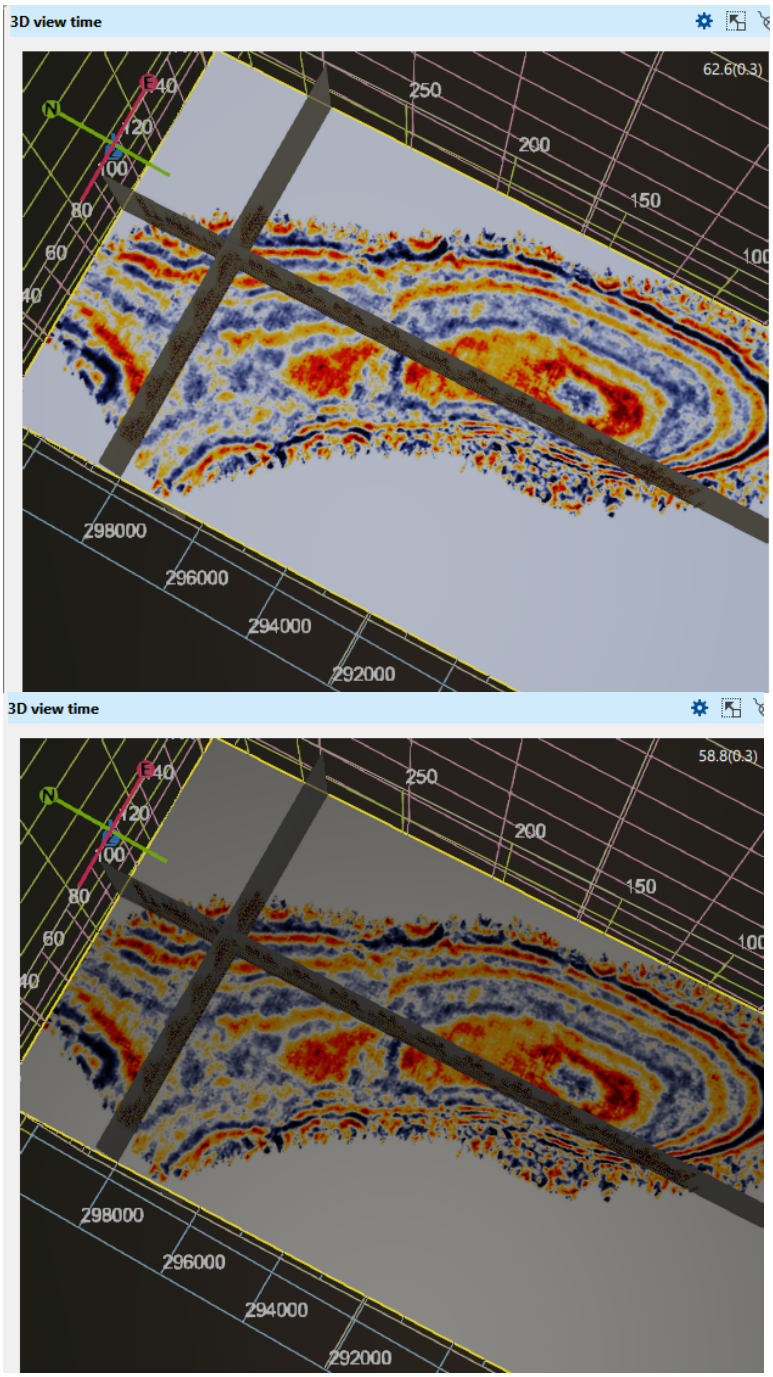
AO AO stands for **Ambient Occulusion**.

Bloom Adds a soft glow around the brightest parts of the scene. By default it is checked; clear it to draw the scene without the glow.

Surface value from Z If the horizons are having attributes (frequency, envelope etc) and the the surface value from Z is turned off, it will take the colorization from the actual attribute value. If the surface value from Z is checked, the colorization will take it from the Z value (time/depth)

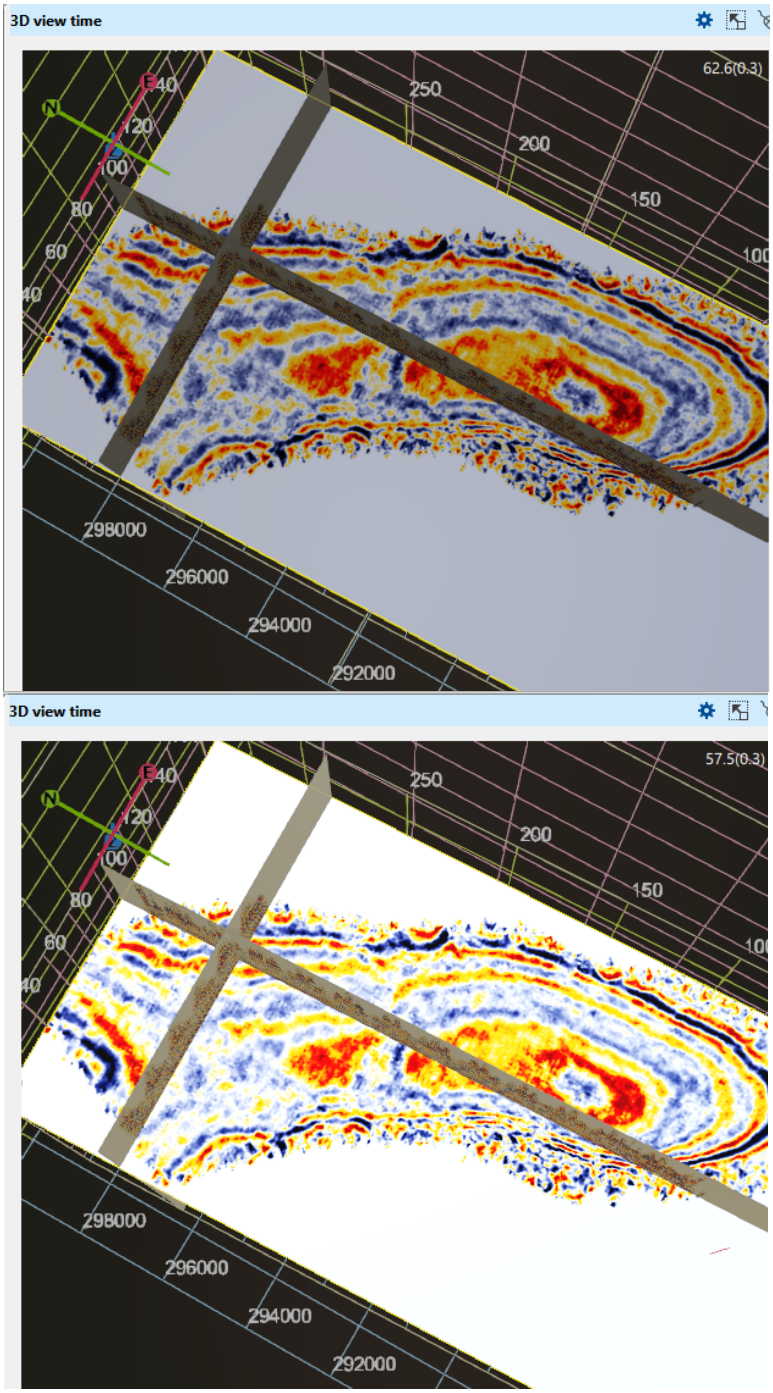
IBL enabled **IBL** stands for **Image Based Lighting (IBL).** By default, it is checked(TRUE). It is also related to the display parameters. When this is disabled, the display looks very less in brightness.

© 2026 Geomage 2003 LTD



IBL multiplier This parameter is directly associated with the **IBL enabled** parameter. By default it is 1. If we change it into say 10, it exponentially increases the brightness.

© 2026 Geomage 2003 LTD

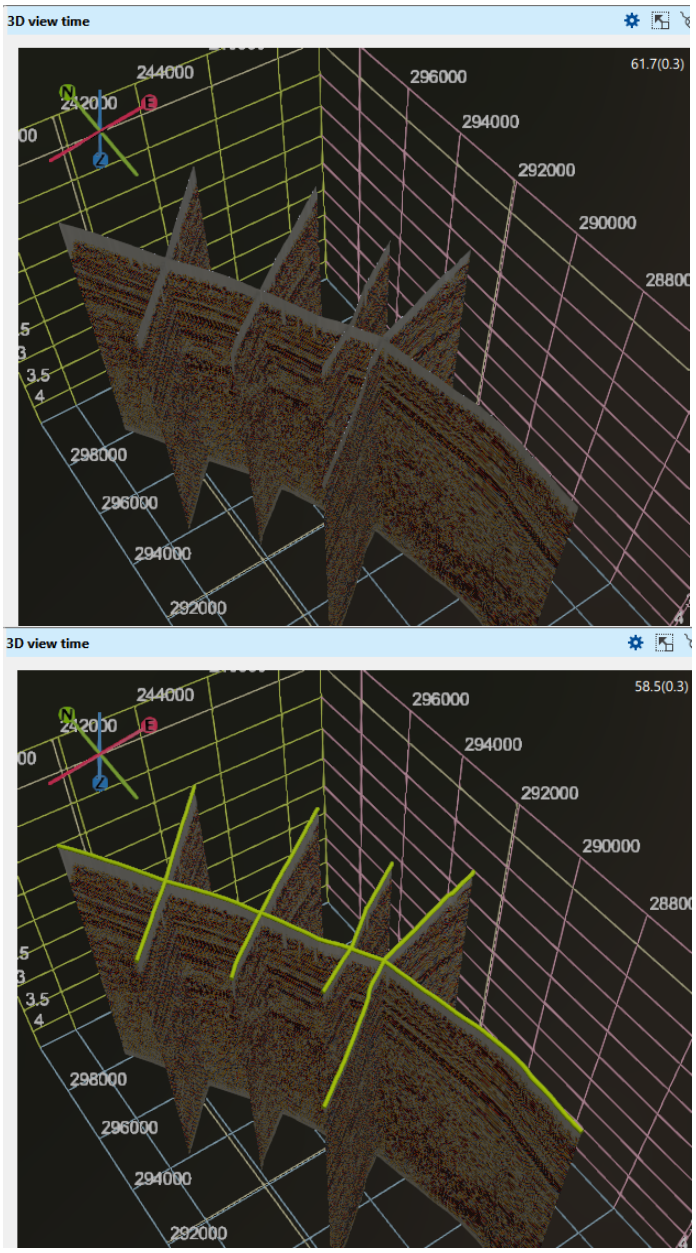


Object(s) tab:

© 2026 Geomage 2003 LTD

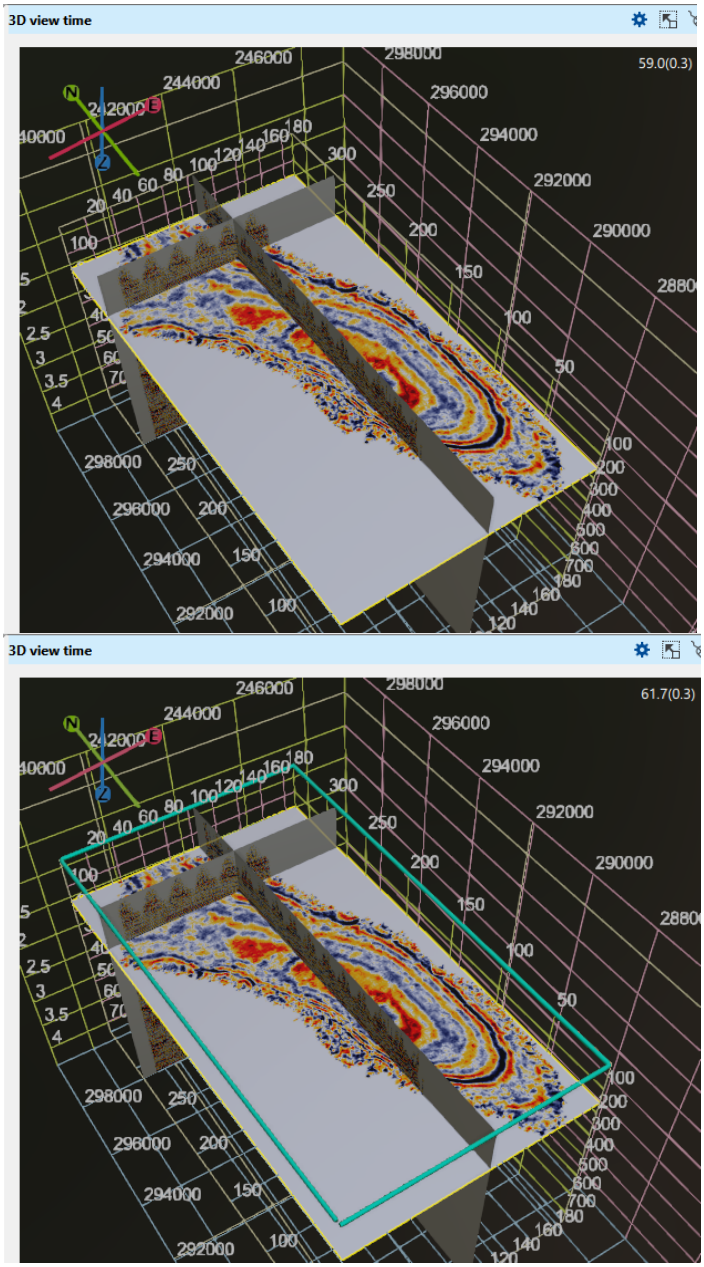
AVO colored bins on AVO crossplot	displaying the calculated AVO parameters and selected
Loaded gathers window	displaying all the loaded 2D lines in the 3D view time
3D volume time	displaying all the loaded 3D volumes
3D Cube attribute time envelope, coherency	displaying any available attributes like frequency,
Surface seismic 2D geometry line boundary	displaying the loaded 2D seismic lines geometry i.e. the

© 2026 Geomage 2003 LTD



Surface seismic 3D geometry It will display the loaded 3D volume boundary like all the corner points.

© 2026 Geomage 2003 LTD

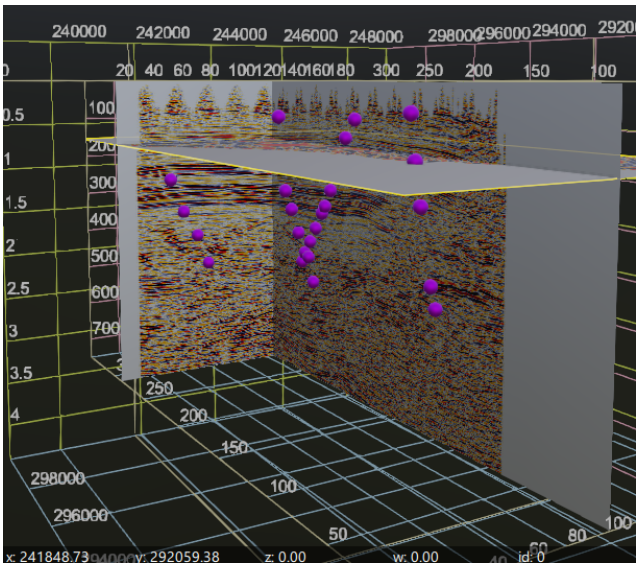


Arbitrary line 3D Displays an arbitrary line

Horizon 3D Topo Displays the topographical horizon

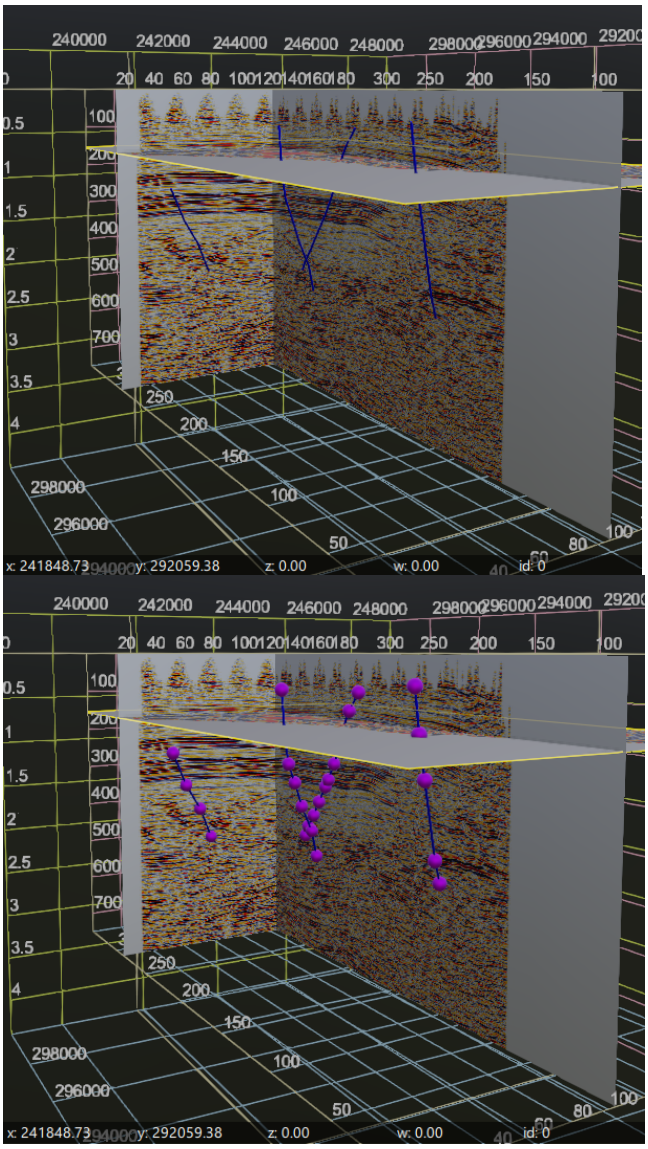
Fault picks This section displays the fault picks. By default, the Object item properties are set to a size of 0.2. Users can adjust this value to view the fault picks in a larger size.

© 2026 Geomage 2003 LTD



Fault sticks 3D This feature functions similarly to the Fault Picks Object item but displays the fault points as a line or stick. It connects the fault points into a continuous line. When both Fault Picks and Fault Sticks are enabled, the display will show a combined view, as illustrated in the right image below.

© 2026 Geomage 2003 LTD



Fault 3D Shows a surface that connects all the sticks associated with the same fault, displaying all the picked or loaded faults in 3D view.

Wells 3D All the loaded wells are displayed here. To manage the visibility of the wells, expand the Wells 3D Object item to check or uncheck them.

Well rig 3D Displays the well rig.

Markers 3D Displays all the well markers**.** The default **Size** value is 1. To display markers for specific wells only, expand the Markers 3D Object item and check or uncheck individual wells, the same way as in the **Data Manager**.

© 2026 Geomage 2003 LTD

Zones 3D Displays each zone of the marker set as a coloured interval drawn along the well trajectory between the zone's top and base marker; a well that is missing one of the two markers shows no interval. To display specific zones only, expand the Zones 3D Object item and check or uncheck them. They are listed as "<zone name> (TVD thickness)" and stay in step with the matching items under **Markers** --> marker set --> **Zones** in the **Data Manager**. Each interval is also labelled from the zone itself, and two Object item properties control that label. **Label content** chooses what it shows - **TVD thickness** (the default), **Zone name** or **Zone name + TVD thickness** - so every well's interval is annotated with its own calculated TVD thickness instead of the zone name repeated at each well. The thickness is written with two decimal places and no unit, for example 309.34; **Zone name + TVD thickness** reads "<zone name>: <value>" and falls back to the zone name alone when no thickness is available, while **TVD thickness** shows nothing for such a well. **Label color** sets the colour of the label text, white by default, independently of the zone colour, which still colours the interval. The labels are drawn only while **Show label** is switched on, which is off by default, and both properties are available for **Zones 3D** in the time and the depth 3D view.

Observing segment Whenever the user selects any bin on the location map, the corresponding position is displayed on the 3D view time window.

Polygon 3D Displays all the loaded polygons**.**

Polygon selection 3D Displays the current/selected or default polygon**.**

Geo-body 3D Displays the geo-bodies stored in the **Geo-bodies** folder of the **Data Manager**. To display specific bodies only, expand the Geo-body 3D Object item and check or uncheck them; each body is listed under it by its **Data Manager** name.

Velocity model A folder with the surfaces of the current velocity model, that is the model set as default in the **Data Manager**. In the time 3D view it holds **Velocity model: interval velocity 3D**, **Velocity model: average velocity 3D** and **Velocity model: time skeleton 3D**; in the depth 3D view it holds **Velocity model: interval velocity 3D**, **Velocity model: average velocity 3D**, **Velocity model: interval-depth skeleton 3D** and **Velocity model: average-depth skeleton 3D**. Each item draws one surface per model layer, at the layer's time map in the time view and at its depth map in the depth view; the velocity items colour the surfaces by interval or average velocity, and the skeleton items by the layer's own time or depth value.

To display specific layers only, expand the Object item and check or uncheck them, or use the check boxes of the same maps under the model in the **Data Manager**: selecting the check box of a group draws all its layers, selecting a single map draws only that layer. The surfaces follow the model that is set as default, so they are rebuilt when you switch the default model or create

© 2026 Geomage 2003 LTD

a new one. Where a map has no values, the surface is left open instead of being drawn across the gap.

Transparency rendering

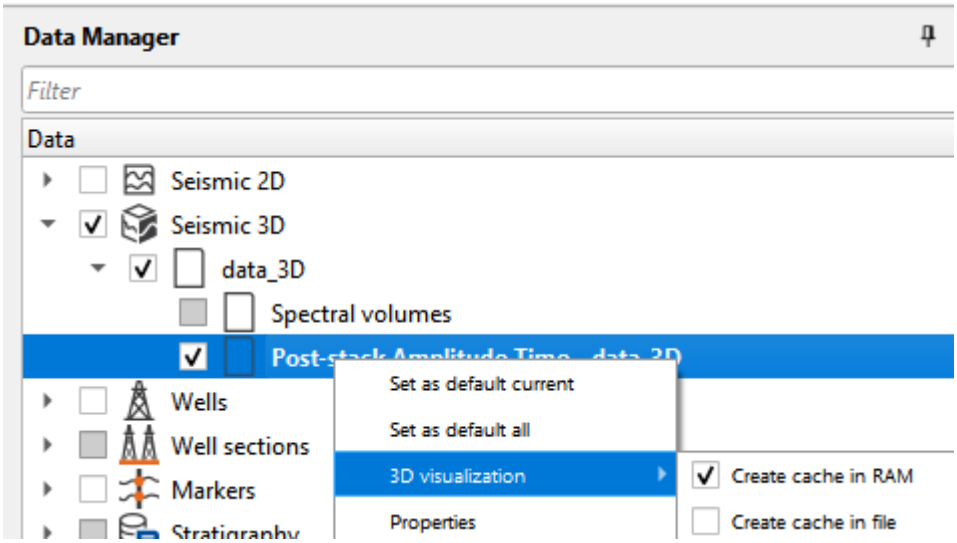
When several transparent objects overlap, the 3D view can render them with two modes, selectable as **Transparency Mode** in the 3D scene settings: **WBOIT** (weighted, blended order-independent transparency) and **PPLL** (per-pixel linked list). PPLL produces a more physically correct, "honest" result where transparent surfaces cross, at a higher rendering cost. When PPLL is selected, the **PPLL Nodes/Pixel** value controls how many overlapping transparent layers are resolved per pixel.

Handling the volume if the memory (RAM) is not enough

If the user is having enough memory for considerable volume size, they can use the option "**Create cache in RAM**". To do that, the user should go to **Data Manager** --> **Seismic 3D** --> **Post-stack Amplitude Time - data_3D** (whatever the name the user has given during the loading of the 3D volume) --> **Right click** --> **3D visualization** --> **Create cache in RAM**. This will use the on board memory to display the 3D volume.

In case the user doesn't have enough memory then they can select the option "**Create cache in file**". This will take little longer time to create cache file when compared to RAM option (depending on the size of the volume, hard disk type (HDD/SSD)). Once the process is completed, now it will access the file from the hard disk and display the 3D volume. The visualization of the 3D volume speed is as good as accessing from the RAM.

© 2026 Geomage 2003 LTD



Hot keys

To rotate the volume in any direction use **MB3** (Right mouse button). User can also able to do the rotation by holding **Ctrl+MB3** button.

To move the volume either up/down or left/right, use **Shift+MB3**(Right mouse button). This option won't rotate the volume. It will remain in it's original position during the movement of volume quite contrast to using **Ctrl+MB3** where it rotates the volume

To move any inline/xline, just click/select on that inline/xline with **MB1** and move wherever you want.

To move the time/depth slice, it's as same as the inline/xline. Simply hold **MB1** and move up/down.

While the **Horizon picks** or **Fault picks** control item is switched on in the view toolbar, **MB1** adds a pick point to the current horizon or fault, and **Ctrl+MB1** does the same. The picking procedures are described in Horizon picking 614 and Fault picking 627.

The user can use the mouse wheel (**MB2** or middle mouse button) to zoom in/out the volume.

To add additional inlines, crosslines click L, to add additional slices for 3D volume click S.

© 2026 Geomage 2003 LTD

3.4.4 AVO views

AVO views group is the set of views where user can visualize pre-stack seismic data and analyze it using graphs and cross-plots

AVO includes following windows:

Image gather 273

AVA gather 276

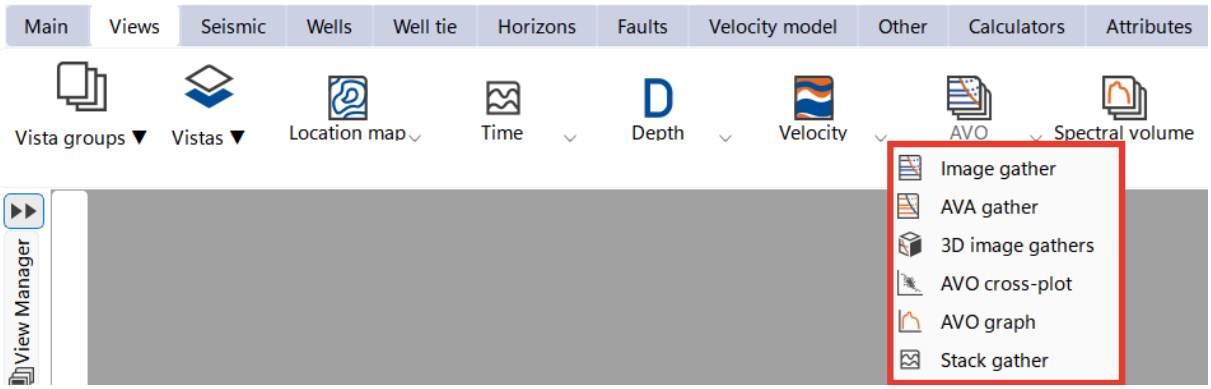
3D image gathers 277

AVO cross-plot 279

AVO graph 280

Stack gather 283

To create AVO section view go to the Views bar 38 on the Ribbon bar 33 and open AVO tab and choose the view that you prefer for data visualization

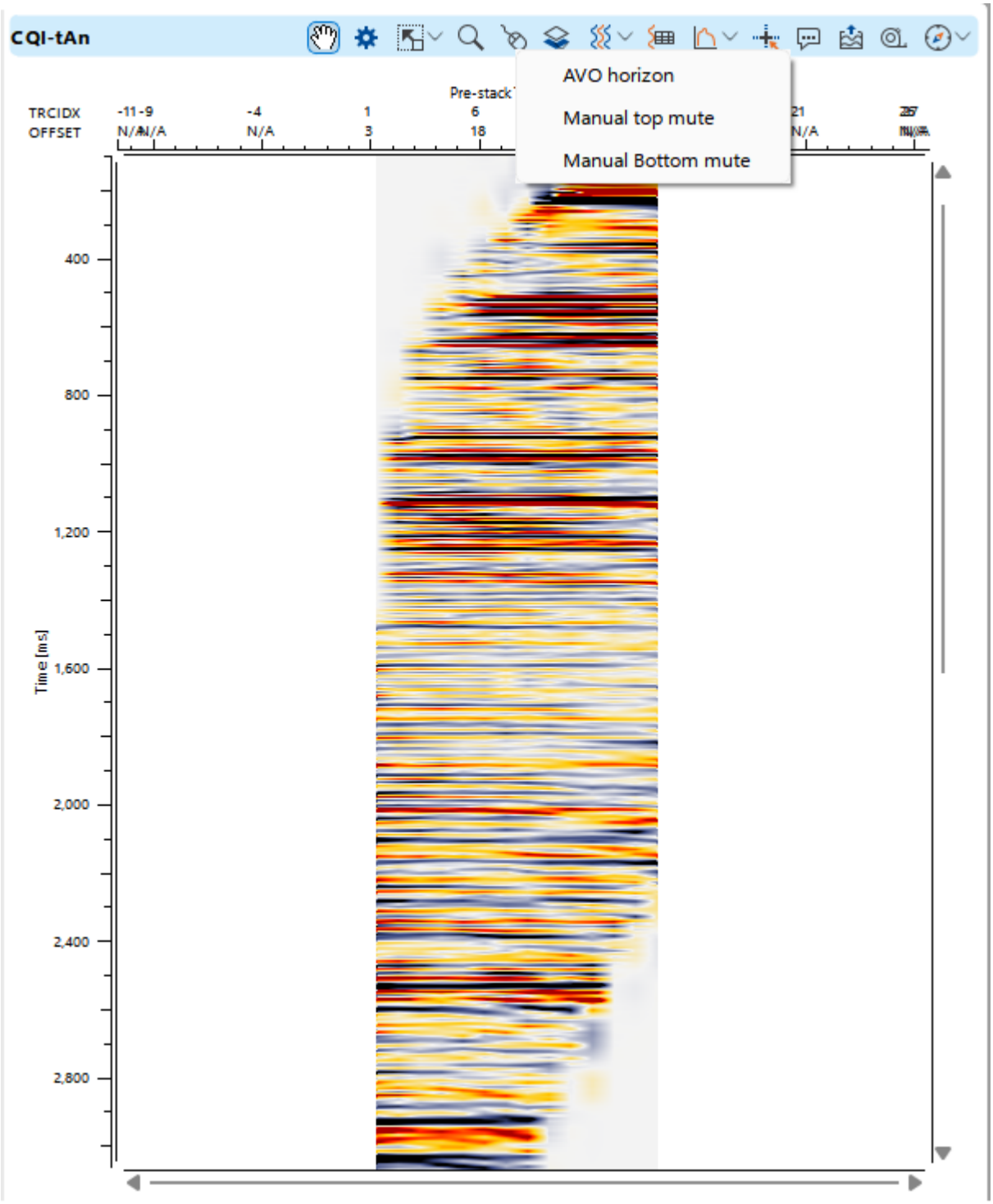


© 2026 Geomage 2003 LTD

3.4.4.1 Image gather

This view displays seismic gathers, showing how amplitude varies with offset or angle for different traces. It is used to analyze AVO behavior across a range of offsets, helping to identify patterns and anomalies that may indicate changes in subsurface properties. The Image Gather View is essential for detailed AVO interpretation, allowing for the examination of amplitude variations before cross-plotting or other analyses.

© 2026 Geomage 2003 LTD



Functional panel allows user to perform the following actions:

open Visual Settings panel

© 2026 Geomage 2003 LTD

adjust view area to all visible objects

set/unset zoomable

set control item from following options:

AVO horizon

Manual Top mute

Manual Bottom mute

add Object

select traces

trace header data

spectral analysis tool - multi selection

spectral analysis tool - single selection

enable mouse tracker

enable tool tip

export picture

ruler tool

show compass

show minimap

show scale bar

© 2026 Geomage 2003 LTD

3.4.4.2 AVA gather

This view displays amplitude versus angle (AVA) data across seismic gathers, allowing for the analysis of how amplitude changes with the angle of incidence. It is useful for identifying variations in subsurface properties, such as lithology or fluid content, by examining the angle-dependent behavior of seismic reflections. The AVA Gather View is crucial for detailed AVA interpretation before further analysis or cross-plotting

Functional panel allows user to perform the following actions:

open Visual Settings panel

adjust view area to all visible objects

set/unset zoomable

set control item

add Object

select traces

trace header data

spectral analysis tool - multi selection

spectral analysis tool - single selection

enable mouse tracker

enable tool tip

export picture

© 2026 Geomage 2003 LTD

ruler tool

show compass

show minimap

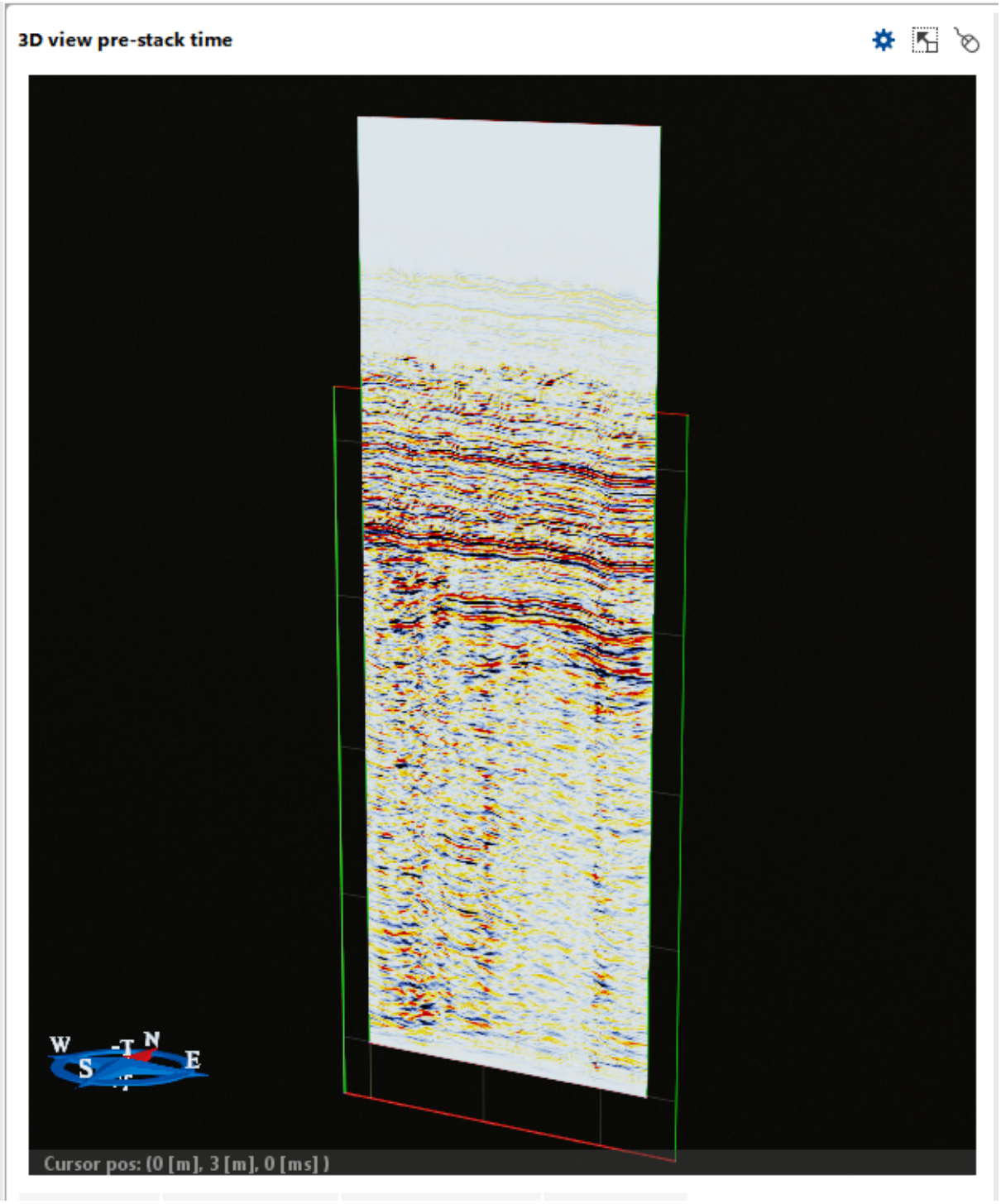
show scale bar

3.4.4.3 3D image gathers

3D image gathers is a window that allows users to operate and visualize any AVO/AVA data in a 3D perspective.

Structurally, the **3D Image Gathers** view shares many settings and functionalities with the general 3D View 244, providing a familiar interface for users. Key customization options include color scaling, amplitude thresholds, transparency adjustments, and the ability to filter by offset or angle ranges, allowing focused analysis of specific features within the dataset. This versatility supports the identification of amplitude anomalies across different offsets and angles, a fundamental aspect of AVO/AVA interpretation.

© 2026 Geomage 2003 LTD

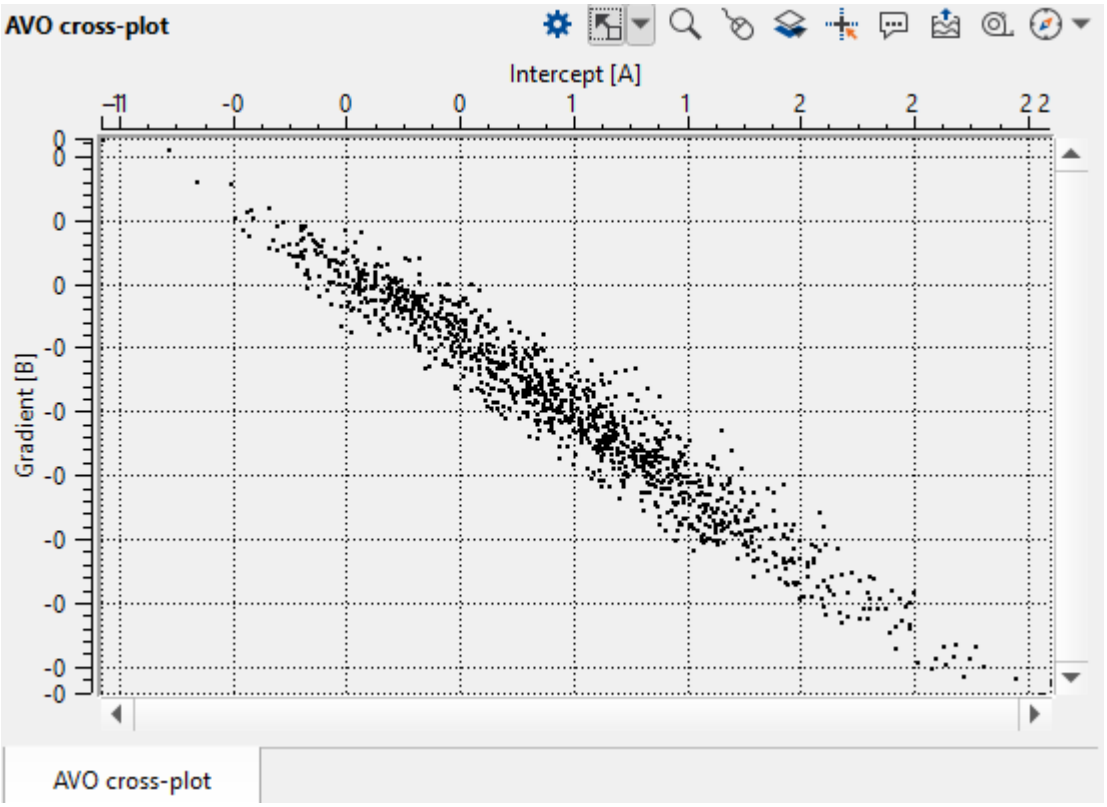


Integration with other views, such as Gather Conditioning 692, enables a comprehensive workflow for assessing and conditioning prestack data. By synchronizing data between views, users can streamline their AVO/AVA analysis, ensuring consistency and enhancing the overall understanding of geological features.

© 2026 Geomage 2003 LTD

3.4.4.4 AVO cross-plot

This scatter plot visualizes AVO data with the x-axis representing the intercept (amplitude at zero offset) and the y-axis representing the gradient (rate of change of amplitude with offset). It is used to quantitatively compare AVO responses from different locations or horizons on a common scale, making it easier to identify data clusters that may indicate specific lithologies or fluid types.



The newly created view will appear in the View manager 189 in the active Workspase 224. User can place AVO cross-plot view by dragging it to any workspace except pre-defined once (Seismic data 228 and Well data 232).

© 2026 Geomage 2003 LTD

To define the object visibility stile go to the Visual Settings 206 by pressing in the view functional panel or find it in the panel 123 sets.

Functional panel allows user to perform the following actions:

open Visual Settings panel

adjust view area to all visible objects

set/unset zoomable

set control item

add Object

enable mouse tracker

enable tool tip

export picture

ruler tool

show compass

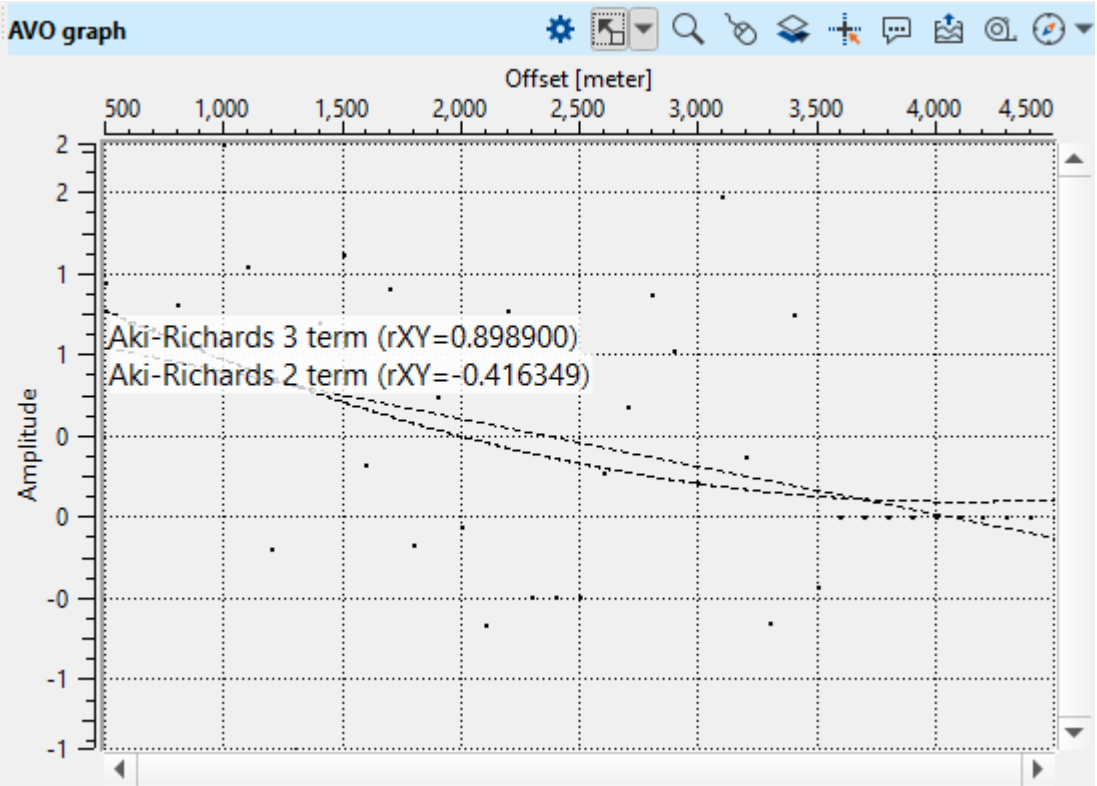
show minimap

 show scale bar

3.4.4.5 AVO graph

This tool graphically represents Amplitude Versus Offset (AVO) data, displaying the relationship between reflection amplitude and offset for selected data points, helping to identify potential hydrocarbon indicators, lithology variations, or fluid content. Users can interactively adjust parameters, filter data points, and apply color-coding or clustering to enhance the analysis.

© 2026 Geomage 2003 LTD



Aki-Richards Approximation Lines

Two approximation curves are overlaid on the data points to model the expected AVO response:

1. **Aki-Richards 2-Term Approximation**: This line uses the first two terms of the Aki-Richards equation, focusing on the intercept and gradient. It provides a simpler approximation that emphasizes the general trend of amplitude variation with offset.
   - **Correlation coefficient (rXY)**: The value displayed here (e.g., -0.416349) represents the correlation between the observed data points and the 2-term approximation, indicating how well this simpler model fits the data.
2. **Aki-Richards 3-Term Approximation**: This line incorporates an additional curvature term, offering a more detailed fit by considering higher-order effects significant at larger offsets.
   - **Correlation coefficient (rXY)**: This value (e.g., 0.898900) indicates the correlation between the observed data and the 3-term model, showing an improved fit with this additional term for higher accuracy in complex cases.

© 2026 Geomage 2003 LTD

Notes

- *Use the 2-term approximation to get a quick estimate of amplitude trends, especially when analyzing data with smaller offsets.*
- *Apply the 3-term approximation for datasets with a broader range of offsets or when high accuracy is required, as it captures more subtle effects related to rock properties and fluid content.*

The newly created view will appear in the View manager 189 in the active Workspase 224. User can place AVO graph view by dragging it to any workspace except pre-defined once (Seismic data 228 and Well data 232).

To define the object visibility stile go to the Visual Settings 206 by pressing in the view functional panel or find it in the panel 123 sets.

Functional panel allows user to perform the following actions:

open Visual Settings panel

adjust view area to all visible objects

set/unset zoomable

set control item

add Object

enable mouse tracker

enable tool tip

export picture

ruler tool

show compass

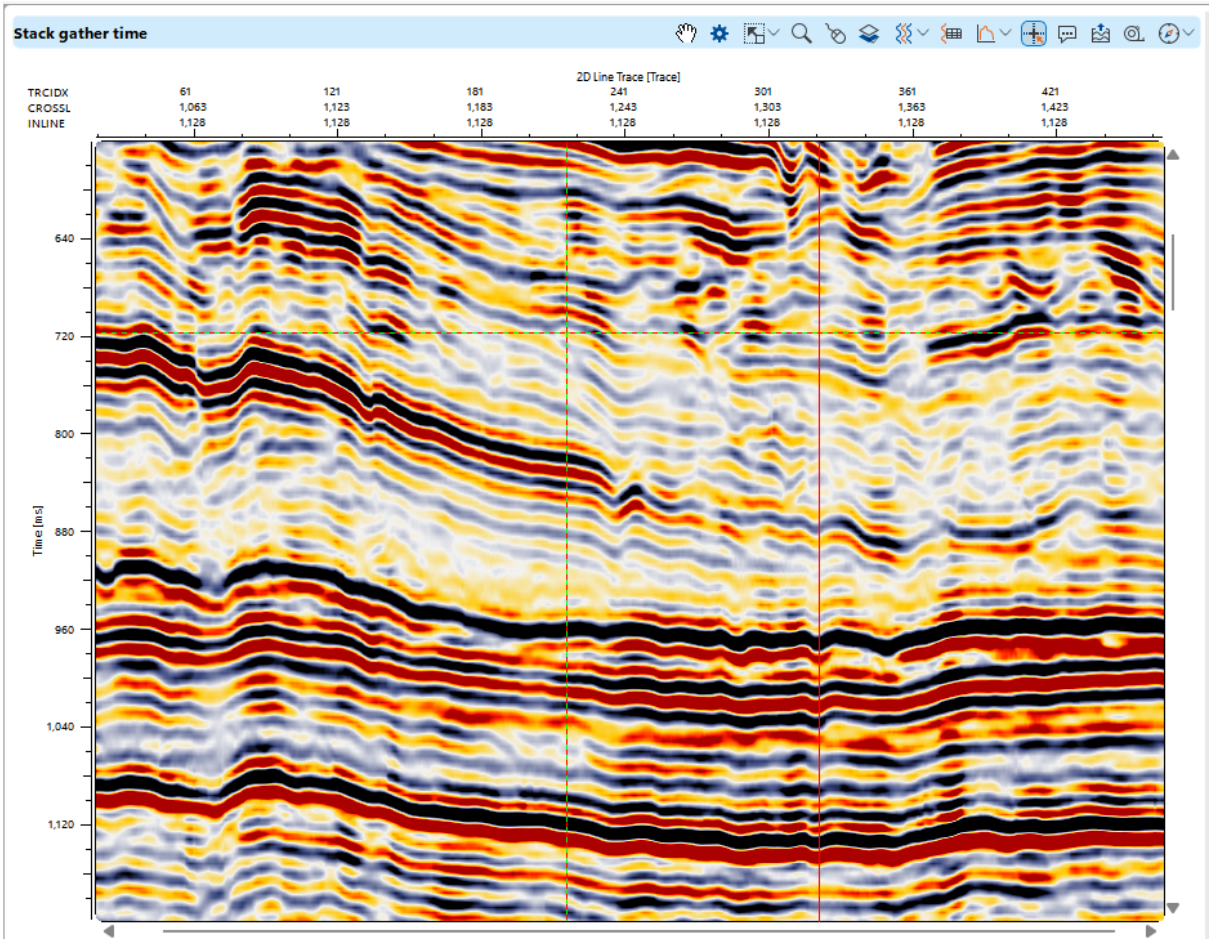
show minimap

© 2026 Geomage 2003 LTD

show scale bar

3.4.4.6 Stack gather

This view presents stacked seismic data over time, combining multiple traces to enhance signal quality and reduce noise. It is used to analyze the summed seismic response at various time intervals, providing a clearer picture of subsurface structures and reflections. The Stack Gather Time View is essential for interpreting the overall seismic response and identifying key geological features.



© 2026 Geomage 2003 LTD

The newly created view will appear in the View manager 189 in the active Workspase 224. User can place Stack gather view by dragging it to any workspace except pre-defined once (Seismic data 228 and Well data 232).

To define the object visibility stile go to the Visual Settings 206 by pressing in the view functional panel or find it in the panel 123 sets.

Functional panel allows user to perform the following actions:

open Visual Settings panel

adjust view area to all visible objects

set/unset zoomable

set control item

add Object

select traces

trace header data

spectral analysis tool - multi selection

spectral analysis tool - single selection

enable mouse tracker

enable tool tip

export picture

ruler tool

show compass

© 2026 Geomage 2003 LTD

show minimap

show scale bar

3.4.5 Table view

This window provides a convenient interface for reviewing and editing various types of data in a table format.

It is particularly useful for managing markers data, log data, seismic attributes, well logs, contacts data, fault data, grid reports and other subsurface information. This feature allows for efficient data updating, comparison, and quality control, making it an essential tool for ensuring the accuracy and consistency of the geo dataset.

Data can be easily sorted by values in descending/ascending or alphabetical order in each column by clicking on the column name, and you can also manually change the values in an editable cell by double-clicking on it.

Markers - 28-AX-34

Filter

Marker	MD (feet)	TVD (feet)	TVDSS (feet)
Shannon_Up	415	415	4769
Niobrara_WS	1962	1962	3222
Carlile_Shale	2434	2434	2750
Frontier_2_WC	3055	3055	2129
Mowry_Shale	3434	3433	1751
Dakota_Sand	3826	3826	1358
Lakota_Sand	3900	3900	1284

Editing data in a Properties table

Objects that store tabular data can be opened as an editable **Properties table**. Add a Table view to the workspace from the Views bar first: the object is loaded into the **Properties table** item of the active Table view, which is not created automatically. Then right-click the object in the Data manager and choose **Show as table**. The caption of the item names the object, for example **Properties table: Inclinometry - <well name>**.

© 2026 Geomage 2003 LTD

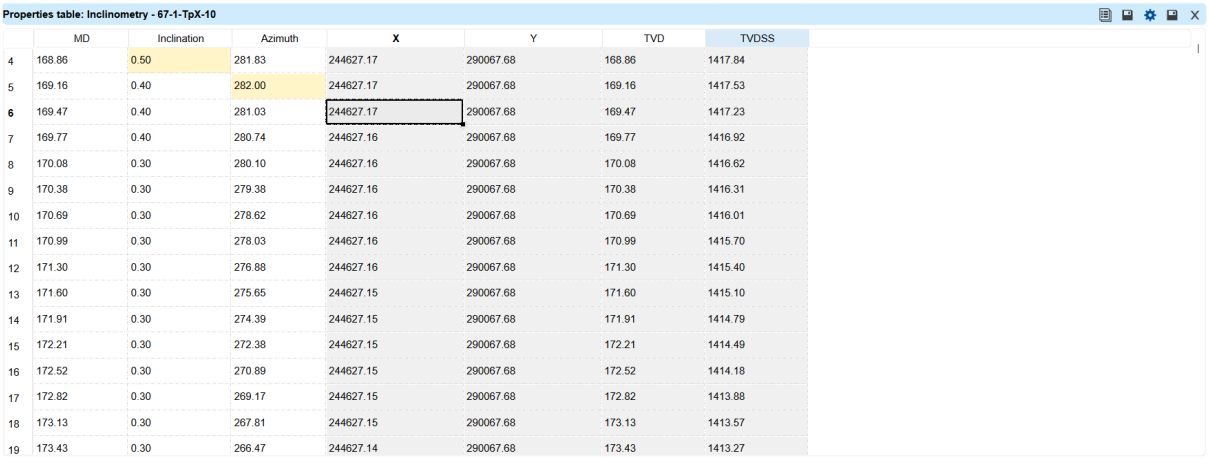
The command is available for:

- a point set under **Points**;
- the **Tops** subsection of a marker set, or a single marker in it;
- the **Zones** subsection of a marker set, or a single zone in it;
- a checkshot of a well;
- the **Inclinometry** subsection of a well.

Only the columns that hold stored values can be changed. The columns that g-Space calculates are read-only and are shown on a grey background:

- **Points** - **X**, **Y** and the attribute columns can be edited; **Point index** is read-only.
- **Marker tops** - **Marker**, **MD**, **Age**, **Type** and the custom attribute columns can be edited; **Well name**, **TVD**, **TVDSS**, **TWT (s)**, **X** and **Y** are read-only. A new value in **Marker** renames that marker in every well of the set, and **Type** accepts **Marker**, **Fault** or **Intrusion**.
- **Zones** - the whole table is read-only. Its values follow the marker tops the zone is built from, so edit those tops instead.
- **Checkshots** - **Base TVD** and **Base OWT (s)** can be edited; **Result TVD** and **Result OWT (s)** are read-only.
- **Inclinometry** - **MD**, **Inclination** and **Azimuth** can be edited; **X**, **Y**, **TVD** and **TVDSS** are read-only.

Double-click an editable cell and type the new value. The value is held in the table and marked on a yellow background; the object itself is not changed yet. For these tables the header of the Table view carries two more buttons, **Save edits** (disk icon) and **Discard edits** (cross icon), which become available as soon as the table holds an unsaved change.

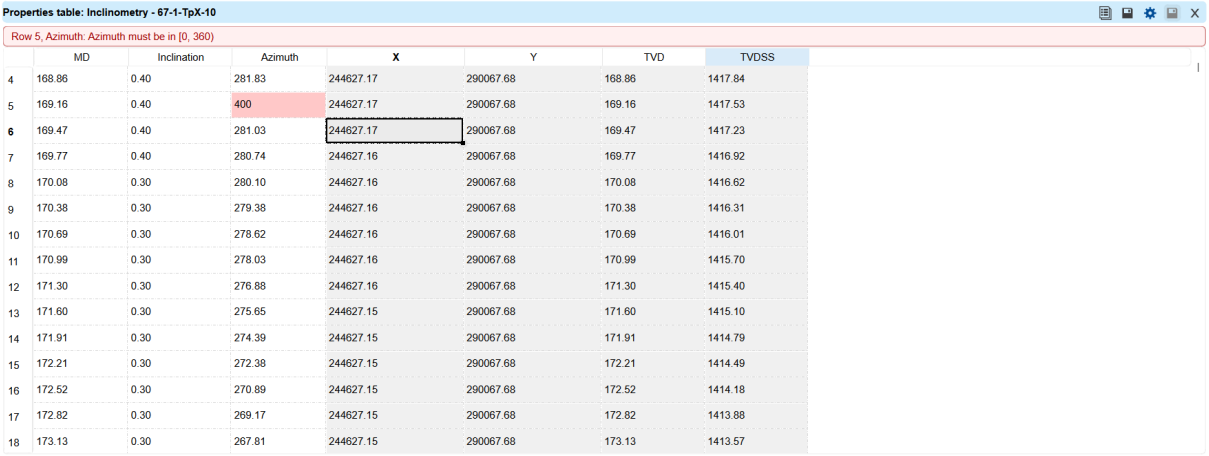


	MD	Inclination	Azimuth	X	Y	TVD	TVDSS
4	168.86	0.50	281.83	244627.17	290067.68	168.86	1417.84
5	169.16	0.40	282.00	244627.17	290067.68	169.16	1417.53
6	169.47	0.40	281.03	244627.17	290067.68	169.47	1417.23
7	169.77	0.40	280.74	244627.16	290067.68	169.77	1416.92
8	170.08	0.30	280.10	244627.16	290067.68	170.08	1416.62
9	170.38	0.30	279.38	244627.16	290067.68	170.38	1416.31
10	170.69	0.30	278.62	244627.16	290067.68	170.69	1416.01
11	170.99	0.30	278.03	244627.16	290067.68	170.99	1415.70
12	171.30	0.30	276.88	244627.16	290067.68	171.30	1415.40
13	171.60	0.30	275.65	244627.15	290067.68	171.60	1415.10
14	171.91	0.30	274.39	244627.15	290067.68	171.91	1414.79
15	172.21	0.30	272.38	244627.15	290067.68	172.21	1414.49
16	172.52	0.30	270.89	244627.15	290067.68	172.52	1414.18
17	172.82	0.30	269.17	244627.15	290067.68	172.82	1413.88
18	173.13	0.30	267.81	244627.15	290067.68	173.13	1413.57
19	173.43	0.30	266.47	244627.14	290067.68	173.43	1413.27

Every entered value is checked immediately. A value that is refused is marked on a red background, the reason is shown in a red line above the table, and **Save edits** stays unavailable until the value is corrected or the changes are discarded. Values must be numeric and inside the range of their column: an inclination between 0 and 180 degrees, an azimuth of 0 or more and

© 2026 Geomage 2003 LTD

less than 360, and depths that stay within the total depth of the well and increase from row to row.



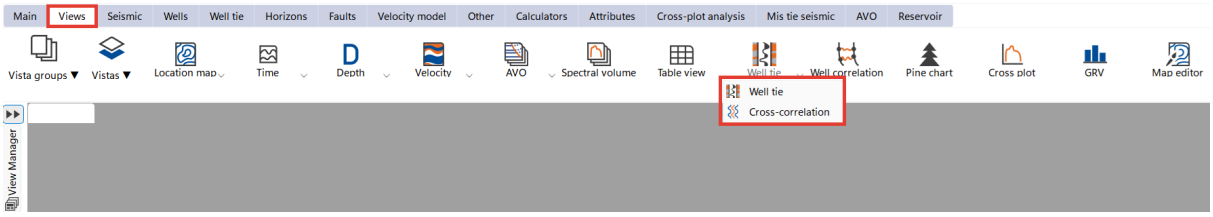
	MD	Inclination	Azimuth	X	Y	TVD	TVDSS
4	168.86	0.40	281.83	244627.17	290067.68	168.86	1417.84
5	169.16	0.40	400	244627.17	290067.68	169.16	1417.53
6	169.47	0.40	281.03	244627.17	290067.68	169.47	1417.23
7	169.77	0.40	280.74	244627.16	290067.68	169.77	1416.92
8	170.08	0.30	280.10	244627.16	290067.68	170.08	1416.62
9	170.38	0.30	279.38	244627.16	290067.68	170.38	1416.31
10	170.69	0.30	278.62	244627.16	290067.68	170.69	1416.01
11	170.99	0.30	278.03	244627.16	290067.68	170.99	1415.70
12	171.30	0.30	276.88	244627.16	290067.68	171.30	1415.40
13	171.60	0.30	275.65	244627.15	290067.68	171.60	1415.10
14	171.91	0.30	274.39	244627.15	290067.68	171.91	1414.79
15	172.21	0.30	272.38	244627.15	290067.68	172.21	1414.49
16	172.52	0.30	270.89	244627.15	290067.68	172.52	1414.18
17	172.82	0.30	269.17	244627.15	290067.68	172.82	1413.88
18	173.13	0.30	267.81	244627.15	290067.68	173.13	1413.57

Save edits writes all held values to the object and the values calculated from them are updated: changed survey stations rebuild the well trajectory, changed checkshot values update the time-depth relationship of the well, and changed point coordinates move the points. The table, the Data manager and the views that show the object are refreshed. **Discard edits** drops the held values and restores the stored ones.

Unsaved changes live only in the table. Loading another object into the Table view discards them without a warning, so save the changes before opening another object.

3.4.6 Well tie view

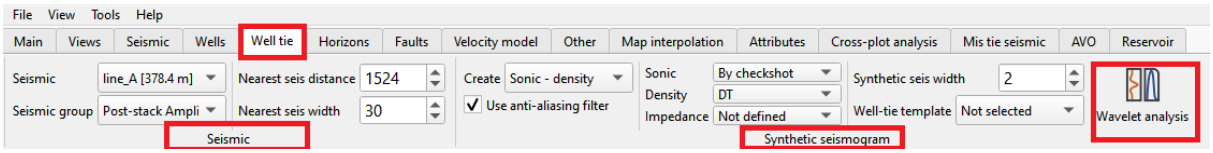
This view is designed for performing, analyzing, and visualizing the well tie process and its results. It consists of two windows: the first displays seismic and well data, while the second shows the cross-correlation function as a numerical outcome of the well tie analysis. This setup allows users to assess the alignment between seismic data and well logs effectively.

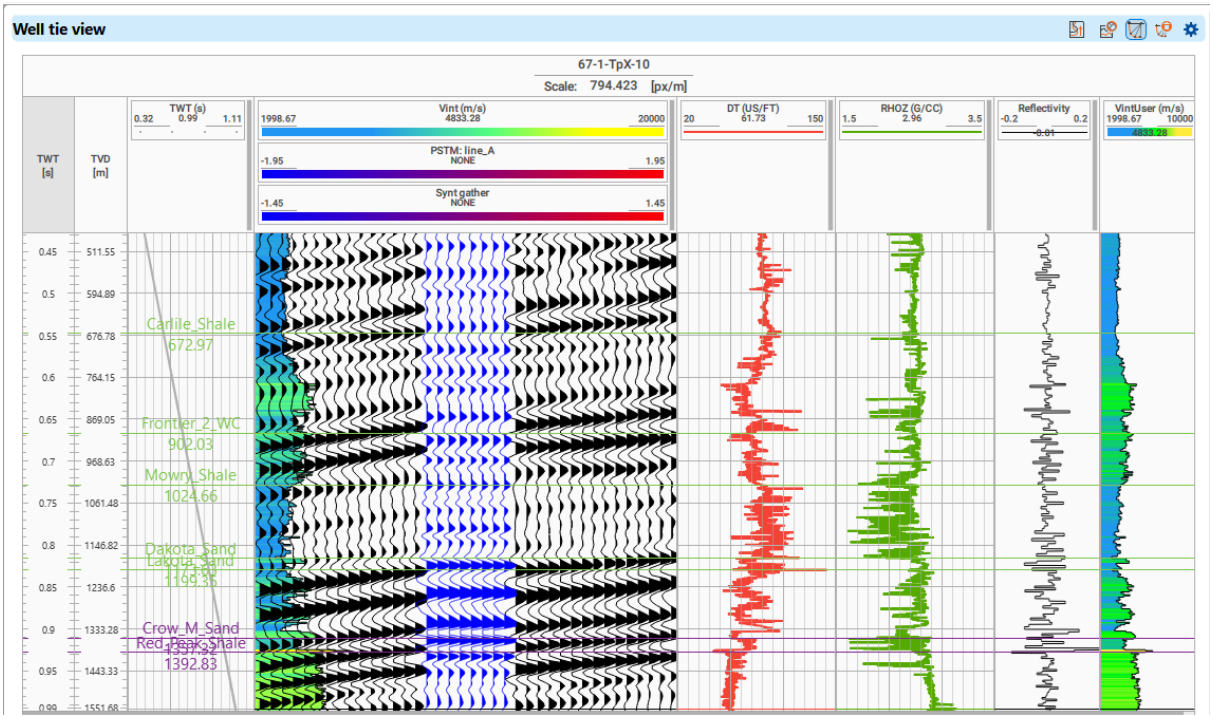


© 2026 Geomage 2003 LTD

Well tie view is a customizable window where users can independently display the data they need for their work, such as log data, time-depth curves, seismic traces, and synthetic traces.

To configure this window, users need to go to the Well Tie bar 93 in the Ribbon bar 33 and select the data required for visualization. The user should have seismic data, well information (check shots/sonic/density/Impedance) to perform the seismic to well tie.





The newly created view will appear in the View manager 189 in the active Workspase 224.

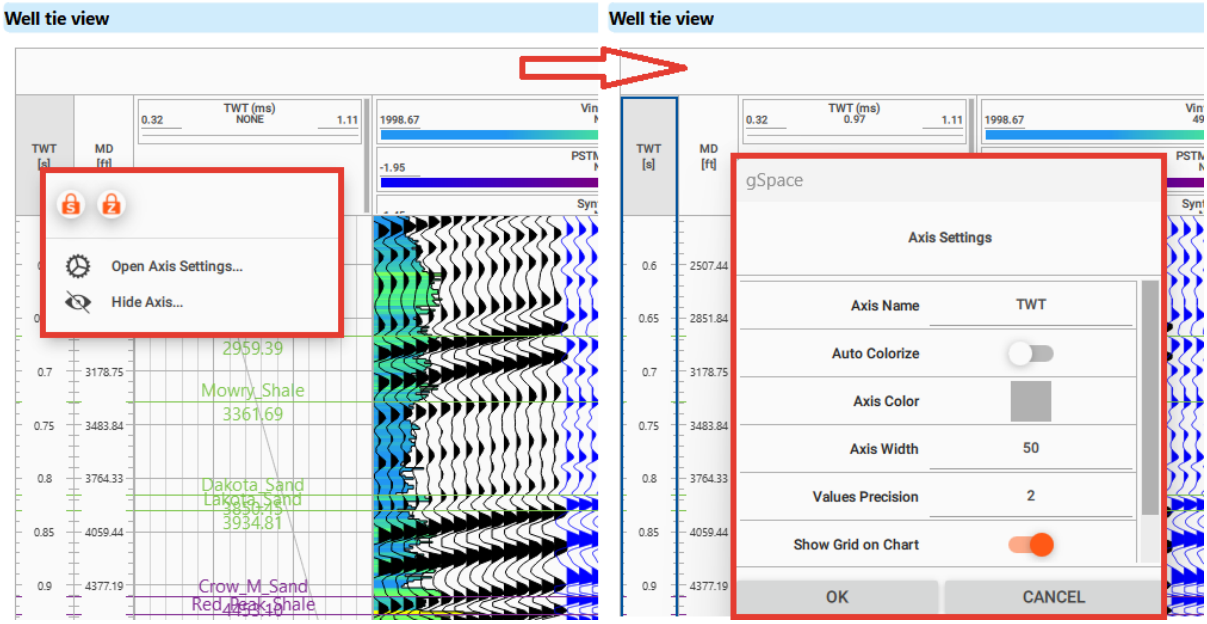
When a well tie view is created, its vertical scale is set from the synthetic seismogram: the interval that carries valid synthetic samples fills the view, with a small margin above and below, so the tied interval is no longer compressed into the full range of the seismic data. Undefined and zero-valued samples at the ends of the synthetic are ignored when this interval is calculated, and a well that has no valid synthetic curve is fitted to its whole content, as before.

© 2026 Geomage 2003 LTD

Zooming and scrolling are not restricted, so the whole data range can still be examined. To return to this default scale after zooming, use **Fit to Synthetic Curve** in the menu that opens when you right-click the field with the well name, described below.

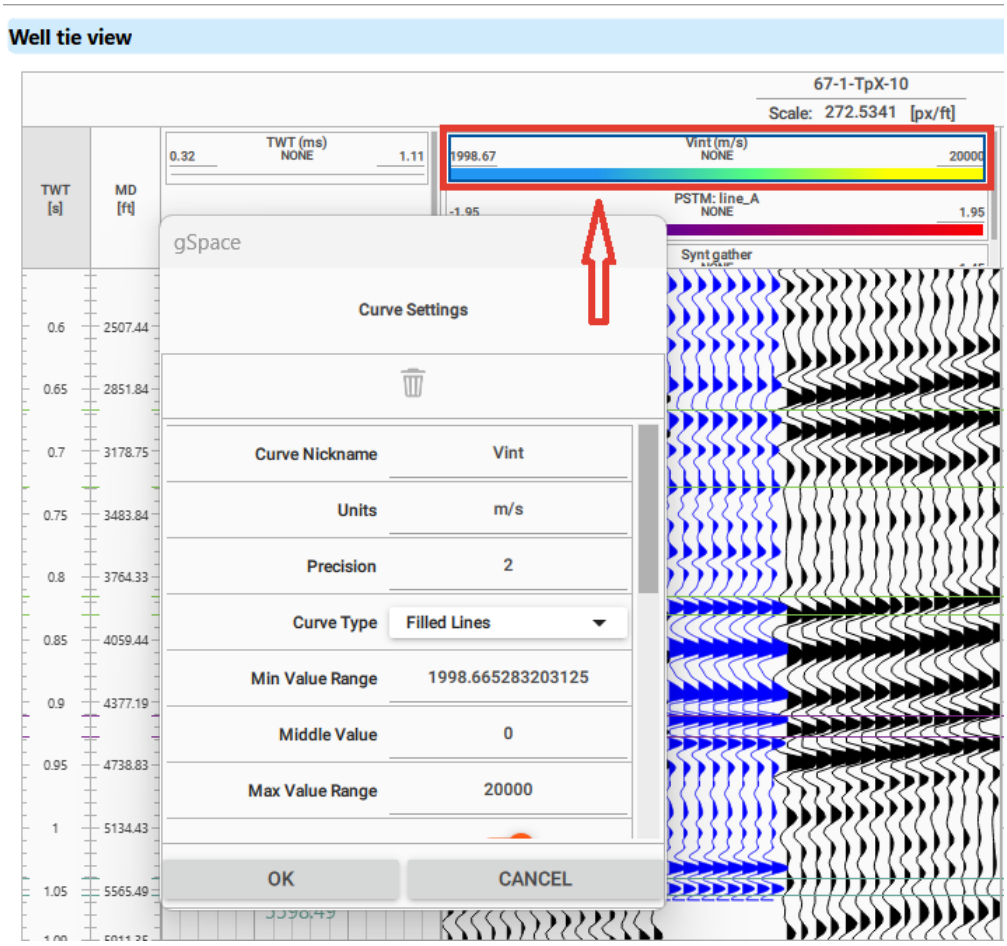
For visualization settings, users can right-click to open the Settings window for the currently active track.

The orange locks with the letters S and Z indicate the scale settings, allowing you to enable or disable scrolling and zooming, respectively, with or without pressing the mouse wheel.



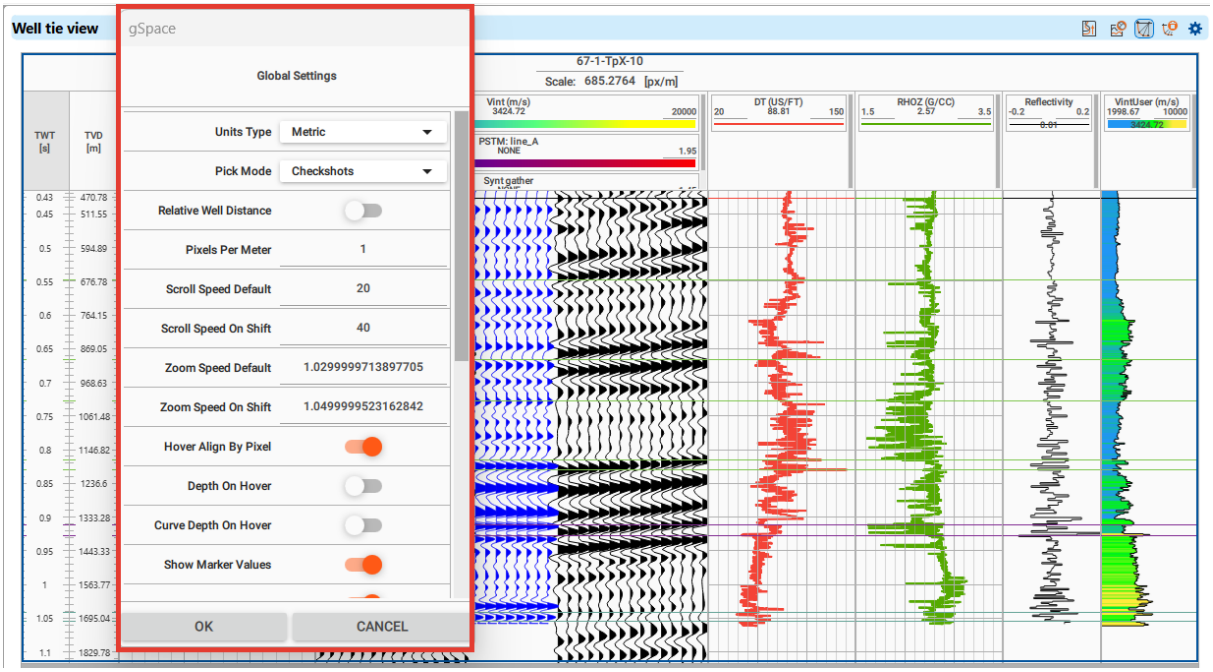
You can easily switch between the settings of each track without closing the settings window by simply clicking on the track you need.
The active track will be highlighted in blue.

© 2026 Geomage 2003 LTD



Clicking on the top field with the well name activates the **Global settings**, which include the following list provided below.

© 2026 Geomage 2003 LTD



You can also access the **Global settings** by right-clicking on the field with the well name, where you will also find the following functionality:

Propagate Well Dimensions Once it is activated, it will propagate the same settings across all the wells.

Fit Well to Content It will fit all the wells to the same. It is kind of Zoomed out.

Fit to Synthetic Curve It scales the view to the interval covered by the synthetic seismogram, the scale a newly created view opens with. A well without a valid synthetic curve is fitted to its whole content instead.

Full size mode It will open a new window for print friendly image to print the well correlation.

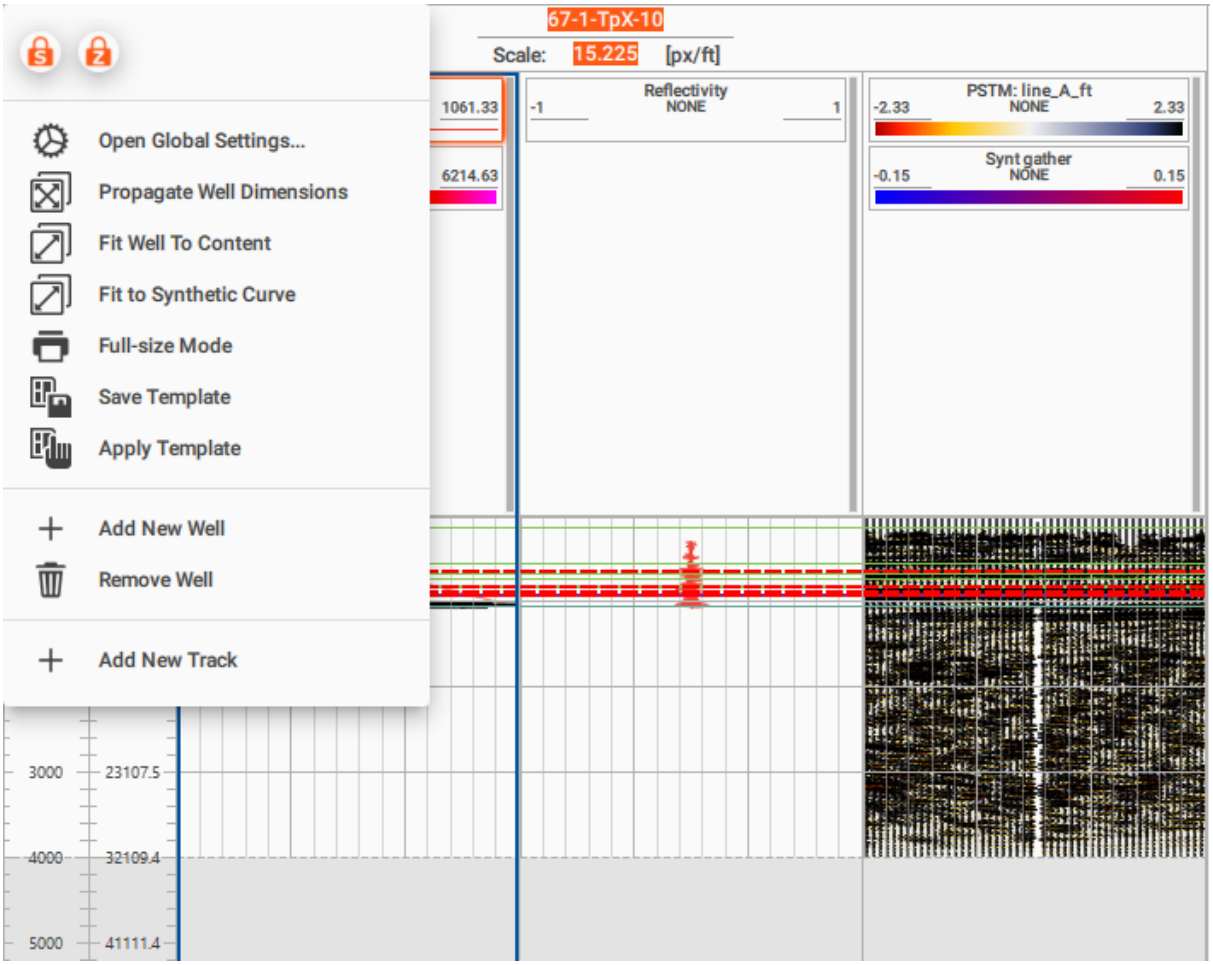
Save Template The user can save the whole setup as a template. You can select a saved template from the Well tie bar 93 under Well Tie Template.

Add New Well This will allow the user to add any additional wells.

Remove Well As the name suggest, the user can remove any wells from the display.

© 2026 Geomage 2003 LTD

Add New Track It will add an empty track



Global settings description

Unit Type The user choose either Metric or Imperial from the drop down menu.

Pick Mode The user choose either Checkshots or Markers

Relative Well Distance By default OFF. If turned ON, it will display the relative well distance

Scroll Speed Default Default scrolling speed settings

© 2026 Geomage 2003 LTD

Scroll Speed on Shift Adjusts scrolling speed

Zoom Speed Default Default zooming speed settings

Zoom Speed on Shift Adjusts zooming speed

Depth on Hover Displays the MD when the cursor is hovered on the well log.

Curve Depth On Hover Displays the curved depth of the well log.

Show Marker Values Displays marker depth values of the well log.

Show Marker Names It will show the marker names of the well log.

Marker Zoom on Hover Enables zooming into markers when the cursor hovers over them.

Propagate Curve data Enables the synchronization of curve data across multiple displays or sections, ensuring consistent updates to related curves when changes are made.

Marker Label Font Sets the font style and size for labels associated with markers.

Axis Font Sets the font style and size for labels associated with axis.

Default Brush Size Defines the default size of the brush tool used for marking or drawing on the display.

Frame Around Lithology Adds a frame around lithology displays.

Linear Velocity Defines the constant velocity used for converting time-based seismic data to depth or distance.

UI COLORS The User Interface (UI) to modify the color settings. When we click on it, it will expand. The user can change the foreground/back ground etc. To collapse it, just click on it one more time. **Anchor Color** sets the colour of the anchor lines that mark the tie points across the tracks; it is red by default.

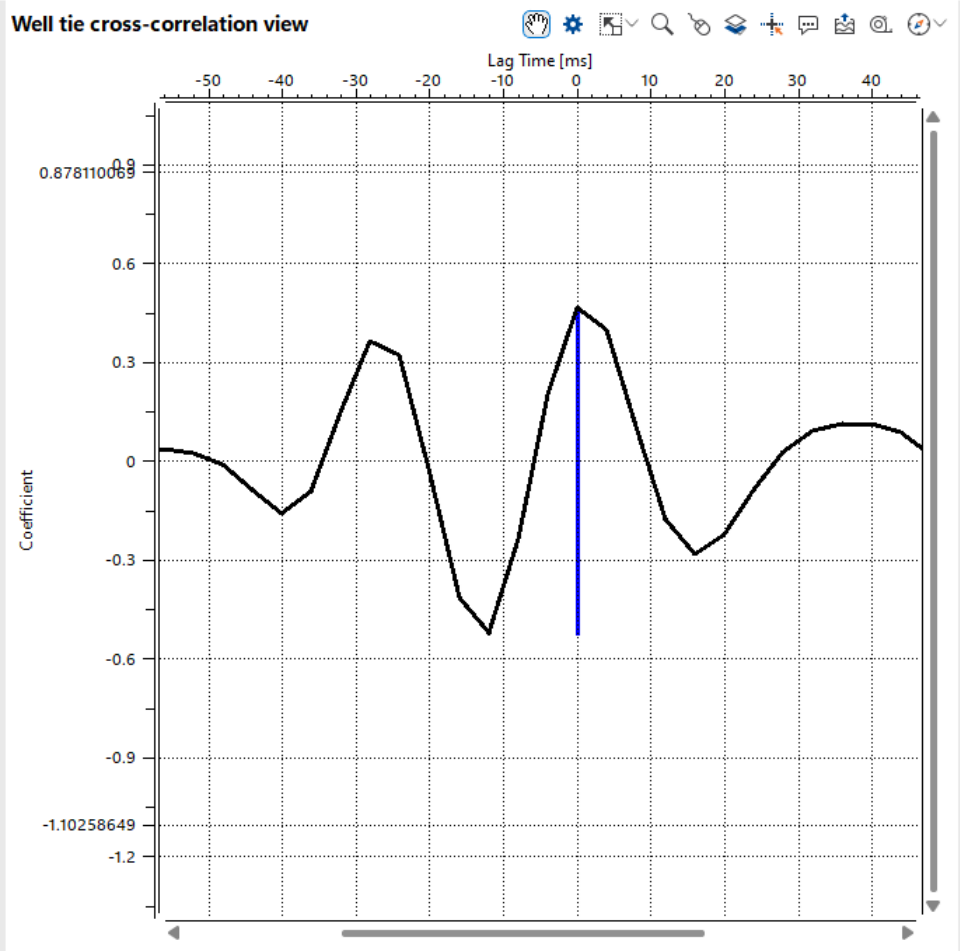
UI DIMENSIONS These settings are related to the Tracks/Horizons/Anchor height and width dimensions. The user can modify as per their choice. **Anchor Width** sets the thickness of those anchor lines, 2 by default.

© 2026 Geomage 2003 LTD

AXIS SETTINGS Inside these settings, the user can ON/OFF the Markers/TWT/TVD etc.

MARKERS VISIBILITY All the loaded markers appear here. The user have the complete control to display which one needed to be displayed.

The **Cross-correlation view** is essential for evaluating the alignment between well log data and seismic data during the well-to-seismic tie process. It displays the cross-correlation function, which quantifies the similarity between the synthetic seismic trace (generated from well logs) and the actual seismic trace. By analyzing the cross-correlation values, users can adjust the time-depth relationship to achieve the best match, ensuring accurate ties between well data and seismic reflections.



Functional panel allows user to perform the following actions:

© 2026 Geomage 2003 LTD

open Visual Settings panel

adjust view area to all visible objects

set/unset zoomable

set control item

add Object

enable mouse tracker

enable tool tip

export picture

ruler tool

show compass

show minimap

show scale bar

Horizons on the seismic track

Horizons kept in the **Horizons** branch of the Data manager 124 are drawn on the well tie track that displays the seismic gather in time. Each horizon appears as a coloured line that follows the reflections trace by trace, the same way a horizon is drawn on a 2D seismic section, instead of a single flat line at the well. Only the track holding the time seismic gather shows the overlay - the synthetic gather and the log tracks are not affected - and only horizons of the time domain are drawn; horizons stored under **Depth** are not shown.

Each horizon keeps the colour assigned to it in the **Data manager**. Where the horizon has no value along the gather, the line breaks instead of being interpolated across the gap. The horizon name is shown next to the line when **Show Marker Names** is enabled in the **Global settings**.

© 2026 Geomage 2003 LTD

To switch the overlay off for a horizon, clear its checkbox in the **Horizons** branch of the **Data manager**, under the user name in the **Time** domain. That horizon is then hidden in the well tie view.

Residual trace

Alongside the synthetic seismogram, a seismic section can display a **Residual seismogram** (residual trace), computed as the difference between the synthetic trace and the seismic trace at the well location (synthetic minus seismic). It is calculated only within the footprint of the synthetic overlay, so its extent matches the synthetic trace window. A flat, low-amplitude residual indicates a good match between the synthetic and the seismic, which makes the residual trace a convenient quality-control display for the tie.

The residual seismogram is generated automatically together with the synthetic trace, in the time domain only. It is a display object of the seismic section views - inline, crossline, arbitrary line and 2D line - rather than a track of the well tie display, so you switch it on in the Object 210 list of the section view. There it is listed next to the synthetic seismogram, named **Residual seismogram** followed by the name of that view, for example **Residual seismogram on inline time**.

Drift curve

When you edit the checkshot anchors, a **Drift** curve - the difference in two-way time (ΔTWT) between the adjusted checkshot and the base checkshot - is generated automatically and shown in the display, and is saved as a LAS curve under the well. See Well Tie 567 for details.

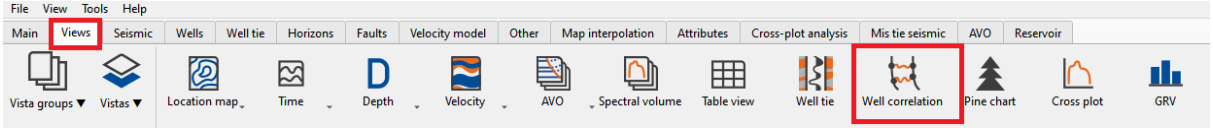
An example of a well-to-seismic tie, along with a detailed description of the required data, can be found in the Best Practice Flows - > Well Tie 567 section

3.4.7 Well correlation view

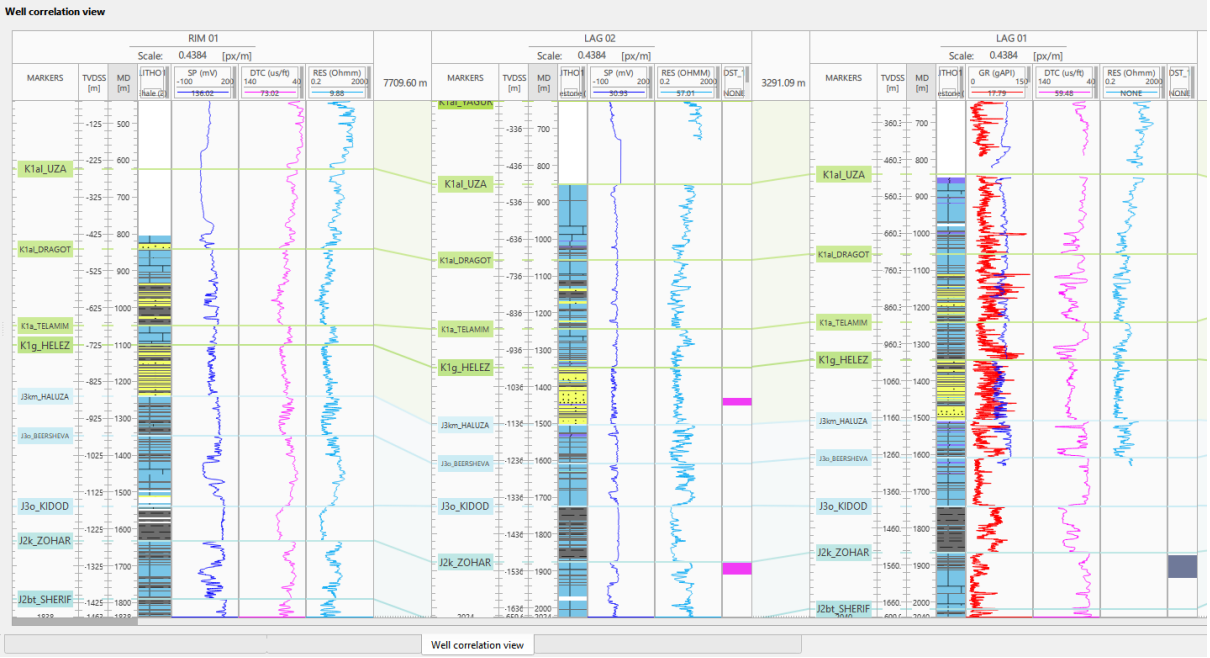
Well correlation - allows the user to create views where the user can visualize the different types of the well data such as log data (continuous and discrete), point data, comment logs etc. Well correlation view is designed to perform easy stratigraphic log correlation.

To create Well correlation view go to the Views bar 38 in the Ribbon bar 33 and click on **Well Correlation** option:

© 2026 Geomage 2003 LTD



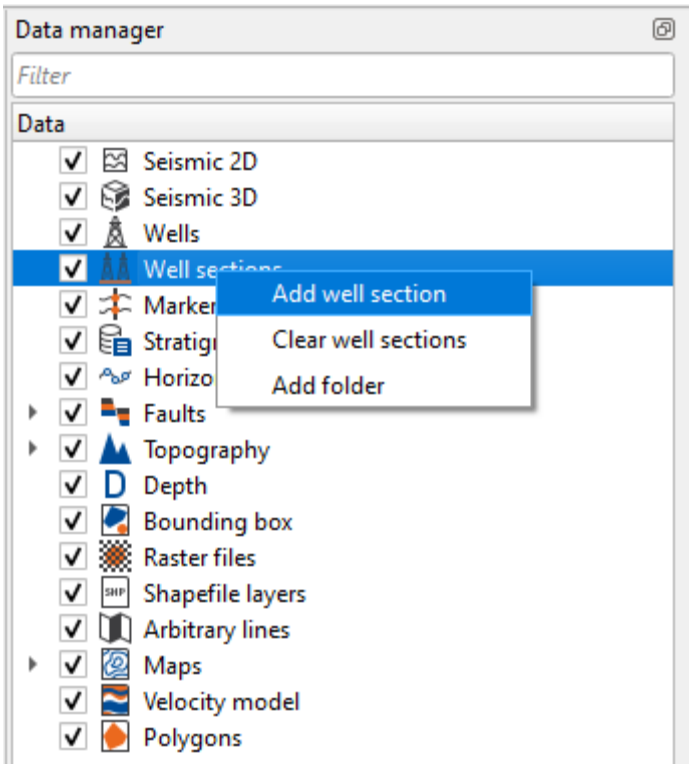
Below is an example display of a well correlation window.



Adding Well sections:

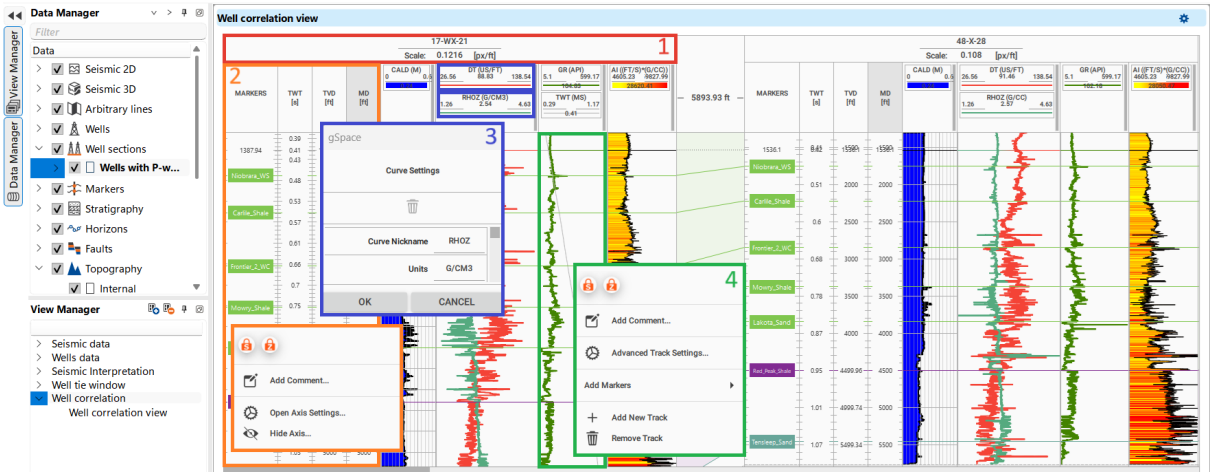
To add a well section, navigate to the Data Manager. Right-click within the interface and select "Add Well Section" from the context menu. A new window will appear prompting you to provide a name for the well section. Enter the desired name and confirm your selection.

© 2026 Geomage 2003 LTD



General settings:

In the following example, we are going to learn how to add/remove wells/tracks/horizons etc for any well inside the well correlation window.



© 2026 Geomage 2003 LTD

The user should be aware of several key functions related to well correlation within the software. The different tracks and displays are highlighted in the picture, and to access their settings, the user can right-click on each section.

Well Name Display (1 - Red): The area at the top of the window highlighted in red shows the well name. Right-clicking on this section allows access to global settings for the well.

A more detailed description of the global settings can be found below.

Axis Tracks and Settings (2 - Orange): The orange-highlighted areas indicate the axis tracks, which display depth, time, or other log data along the vertical axis.

By right-clicking on these tracks, users can access settings to adjust the scale or other visualization options specific to the axis.

The vertical axis can be zoomed in or out by placing the cursor within the axis track and adjusting the scale accordingly. In contrast, the horizontal axis can be scrolled from right to left and vice versa.

To set the priority scale, first, visualize the desired axis, then right-click to open its settings, and make it active.

The title of the active axis will turn gray, indicating it has been prioritized. By default, the priority axis is set to measured depth (MD), as shown in the figure where the MD field is highlighted in gray.

When the comments column is displayed (see **Show Comments** in the Global Settings described below), right-clicking that column opens the same **Axis Settings** window with four additional rows that apply to the comments. These rows are not shown on depth or time axes such as MD, TVD or TWT.

Font Size: Size of the comment text in the comments column, 10 by default. The pop-up that appears when the cursor is hovered over a comment to read its full text uses the same size.

Comment Color: Color of the comment text, black by default.

Show MD: By default OFF. If turned ON, the measured depth of each comment is added in parentheses after the comment text, for example Top Sand (1520), while a comment with no text shows the depth on its own. The depth follows the view units and the **Values Precision** set on the same axis.

© 2026 Geomage 2003 LTD

Wrap Text: By default ON. Long comment text is wrapped over several lines so that it stays readable when the comments column is narrow, and the comment row grows in height to fit. Turn it off to keep each comment on a single line.

Curves Tracks and Settings (3 - Blue): The blue-highlighted sections show the individual curves or log tracks, such as GR (Gamma Ray), DT (Sonic), or AI (Acoustic Impedance).

Users can access the curve settings by right-clicking within this area, as shown in the inset labeled "3." Here, they can adjust parameters like curve nickname or units.

Tracks Displays and Settings (4 - Green): The green-highlighted areas indicate where users can manage the display of individual tracks. By right-clicking here, the user can add comments, access advanced track settings, add curves, as well as add new tracks and remove them. The width of a track can also be changed here directly: press the left mouse button on its right border and drag horizontally.

To access Global Settings, the user should right-click inside the Well Name area (highlighted in red) as shown below.

Additional tools are also available.

© 2026 Geomage 2003 LTD

Open Global Settings It is where the user can add/remove any well markers/curves etc besides changing the overall look and feel of the display.

Propagate Well Dimensions Once it is activated, it will propagate the same settings across all the wells.

Fit Well to Content It will fit all the wells to the same. It is kind of Zoomed out.

Fit to Synthetic Curve It scales the depth axis of the well to the interval covered by its synthetic seismogram. A well that has no synthetic is fitted to its full content instead.

Full-size Mode It will open a new window for print friendly image to print the well correlation.

Save Template The user can save the whole setup as a template.

Apply Template Apply a previously saved template to this well.

Add New Well This will allow the user to add any additional wells.

Remove Well As the name suggest, the user can remove any wells from the display.

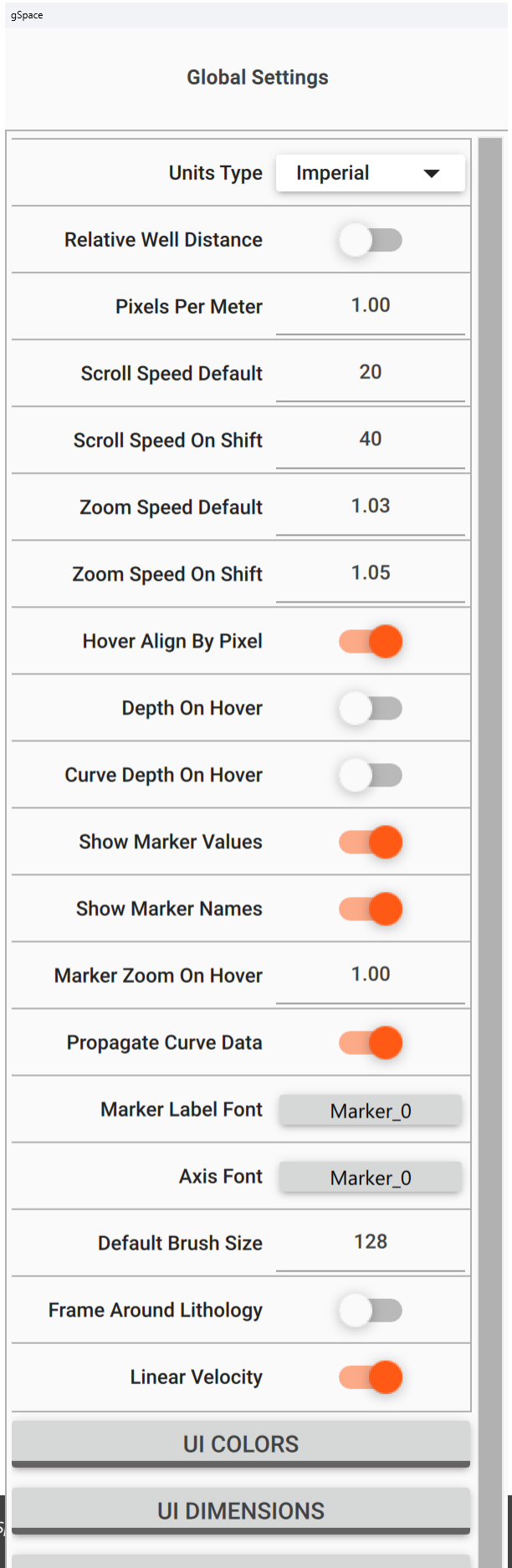
Add New Track It will add an empty track

Well correlation views and well sections keep their current display settings when you switch away and return. Saving the project also saves each current state, so reopening restores it even if you did not save the settings as a template. If you reopen without saving the latest display changes, those changes are discarded; a well section with an assigned template returns to that template.

Out of these all options, the most important one is **Open Global Settings**. We are going to learn about it in the following section.

Select **Open Global Settings** and a new pop-up window opens.

© 2026 Geomage 2003 LTD



© 2026 Geomage 2003 LTD

Units Type The user choose either Metric or Imperial from the drop down menu.

Relative Well Distance By default OFF. If turned ON, it will display the relative well distance

Depth on Hover It will display the MD when the cursor is hovered on the well log.

Curve Depth On Hover Displays the curved depth of the well log.

Show Marker Values Displays marker depth values of the well log.

Show Marker Names It will show the marker names of the well log.

Linear Velocity

UI COLORS This is nothing but User Interface (UI) to modify the color settings of the displays. When we click on it, it will expand.The user can change the foreground/back ground etc.To collapse it, just click on it one more time.

UI DIMENSIONS These settings are related to the Tracks/Horizons/Anchor height and width dimensions. The user can modify as per their choice.

AXIS SETTINGS Click this group to expand it. **Show TVD**, **Show TVDSS**, **Show OWT**, **Show TWT** and **Show Markers** show or hide the matching column next to the depth axis. The group also holds the two comment options below.

Show Comments Shows or hides the separate comments column next to the depth axis.

Show Comments on Curves By default OFF. If turned ON, every well comment is drawn at its depth as a horizontal line in the color of that comment across the curve tracks, and the comment text appears above the line in the first track. The two options are independent, so comments can be shown in their own column, directly over the curves, or both.

Markers Visibility All the loaded markers appear here. The user have the complete control to display which horizons needed to be displayed.

© 2026 Geomage 2003 LTD

Map: <map name> Beside Markers Visibility, one such group is added for every depth map that is currently displayed on the view. Its caption is the name of the map, followed by the owning user in parentheses when the map has one. The group holds **Visibility**, **Color**, **Line Style**, **Line Thickness** and **Labels**, which draws the name of the map next to its line. See Depth maps below.

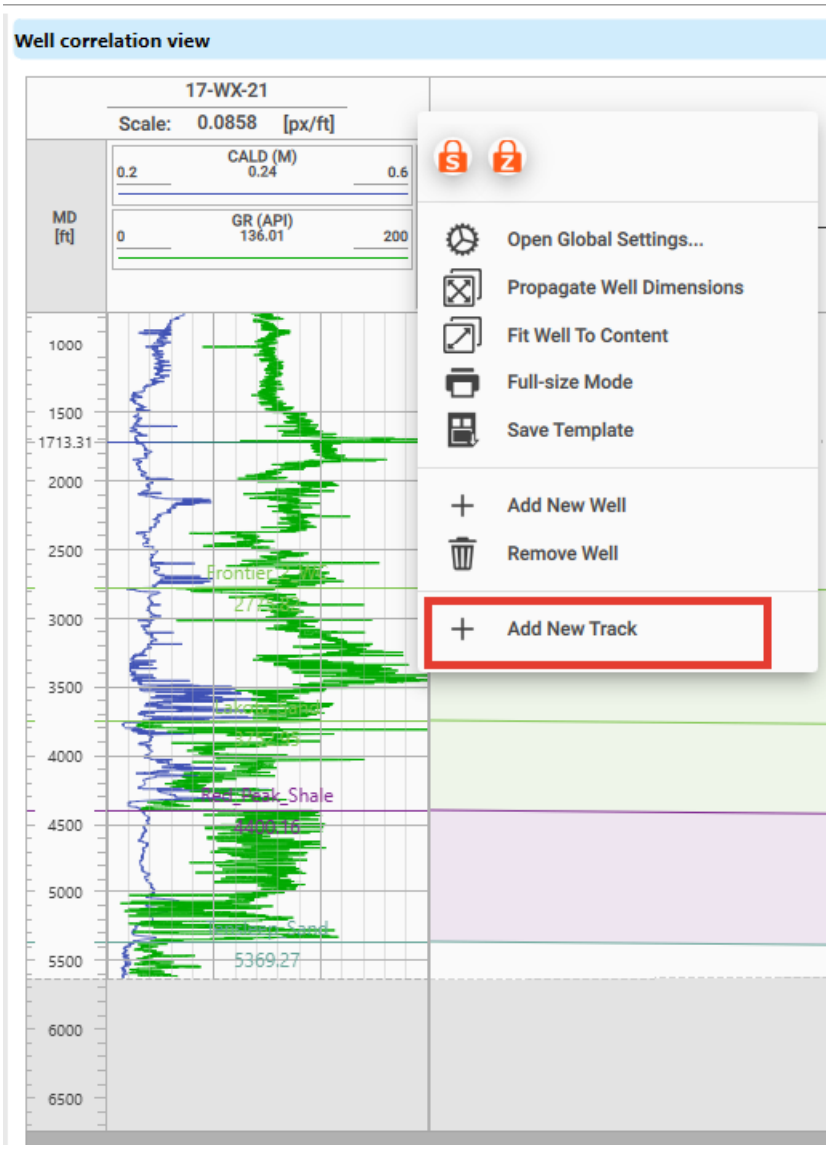
There isn't a single, universally accepted international color palette for log curves, but certain conventions are commonly followed in the industry. Here are some typical color associations for log curves:

1. *Gamma Ray (GR): Green*
2. *Resistivity (ILD, LLD, etc.): Red*
3. *Density (RHOB): Brown*
4. *Neutron Porosity (NPHI): Blue*
5. *Sonic/Acoustic (DT): Black or Blue*
6. *SP (Spontaneous Potential): Black*
7. *Porosity: Blue for neutron porosity, yellow for density porosity*
8. *Caliper: Blue or Purple*

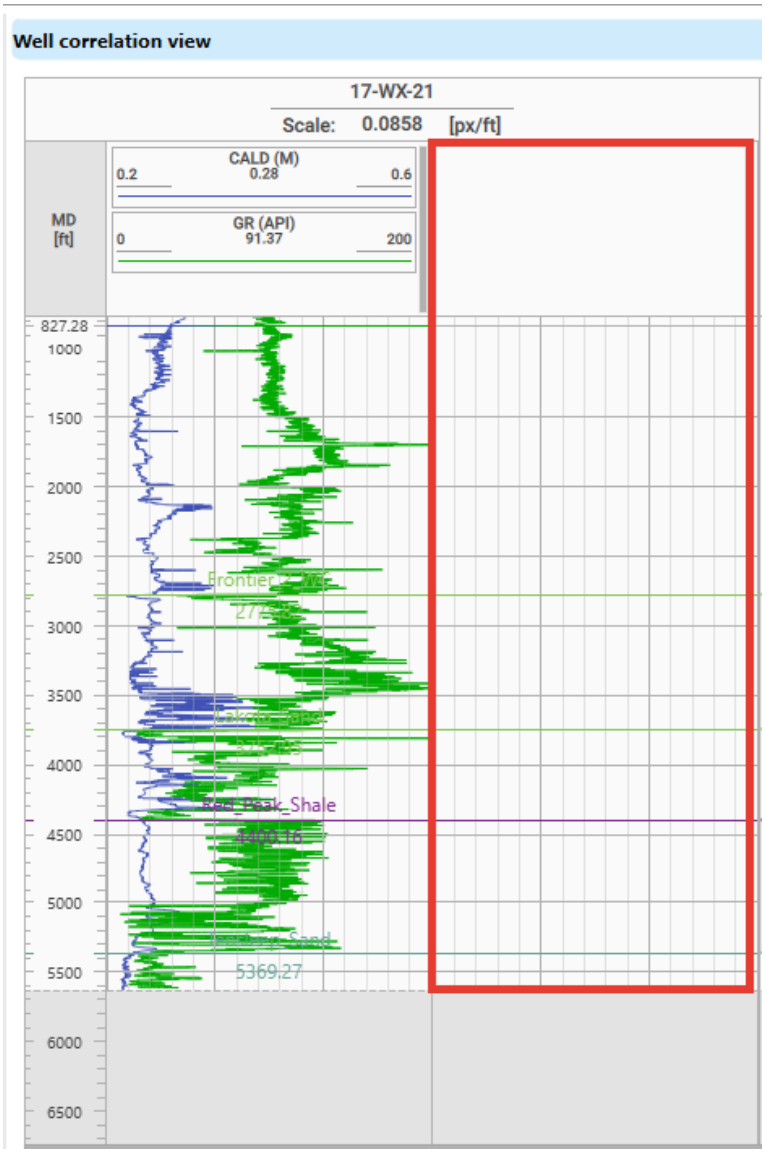
Adding new track, well logs:

To add new track, **Right click** inside the **Well name** as shown below and select "**Add new track.** It will add an additional panel (Green rectangle) to the existing one as shown below.

© 2026 Geomage 2003 LTD

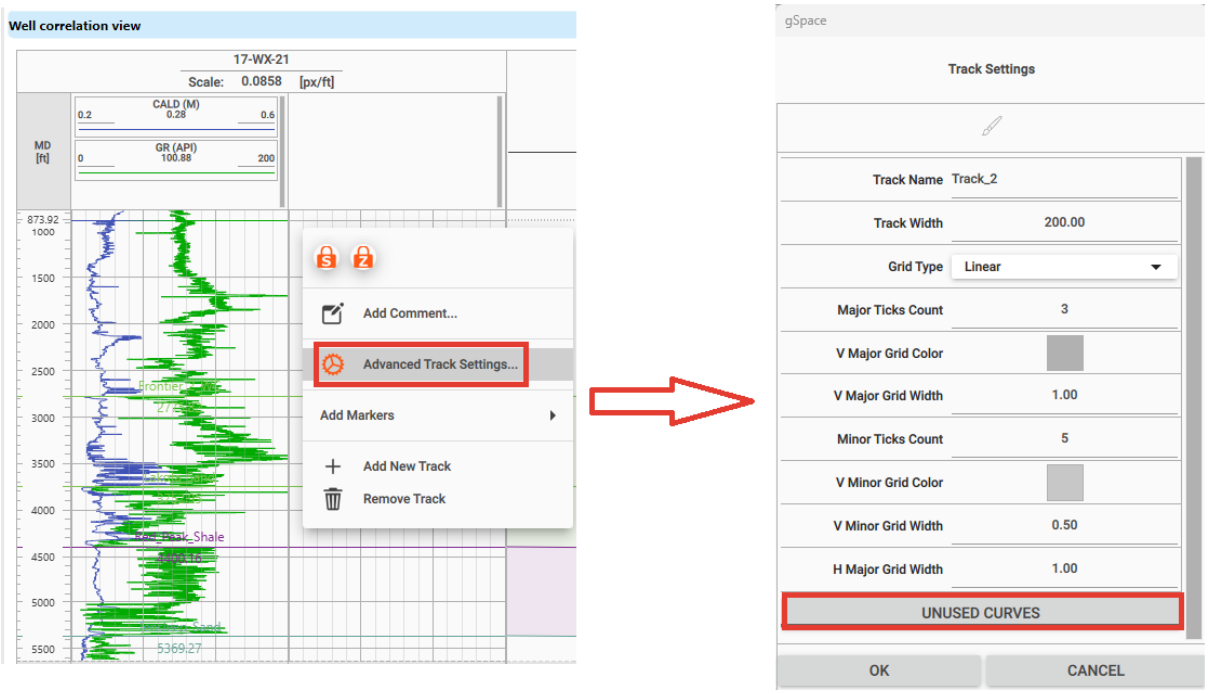


© 2026 Geomage 2003 LTD



To add any well logs inside this, the user should **Right click** inside the **New Track window** and select **Advanced Track Settings**. It will open a new window as shown below.

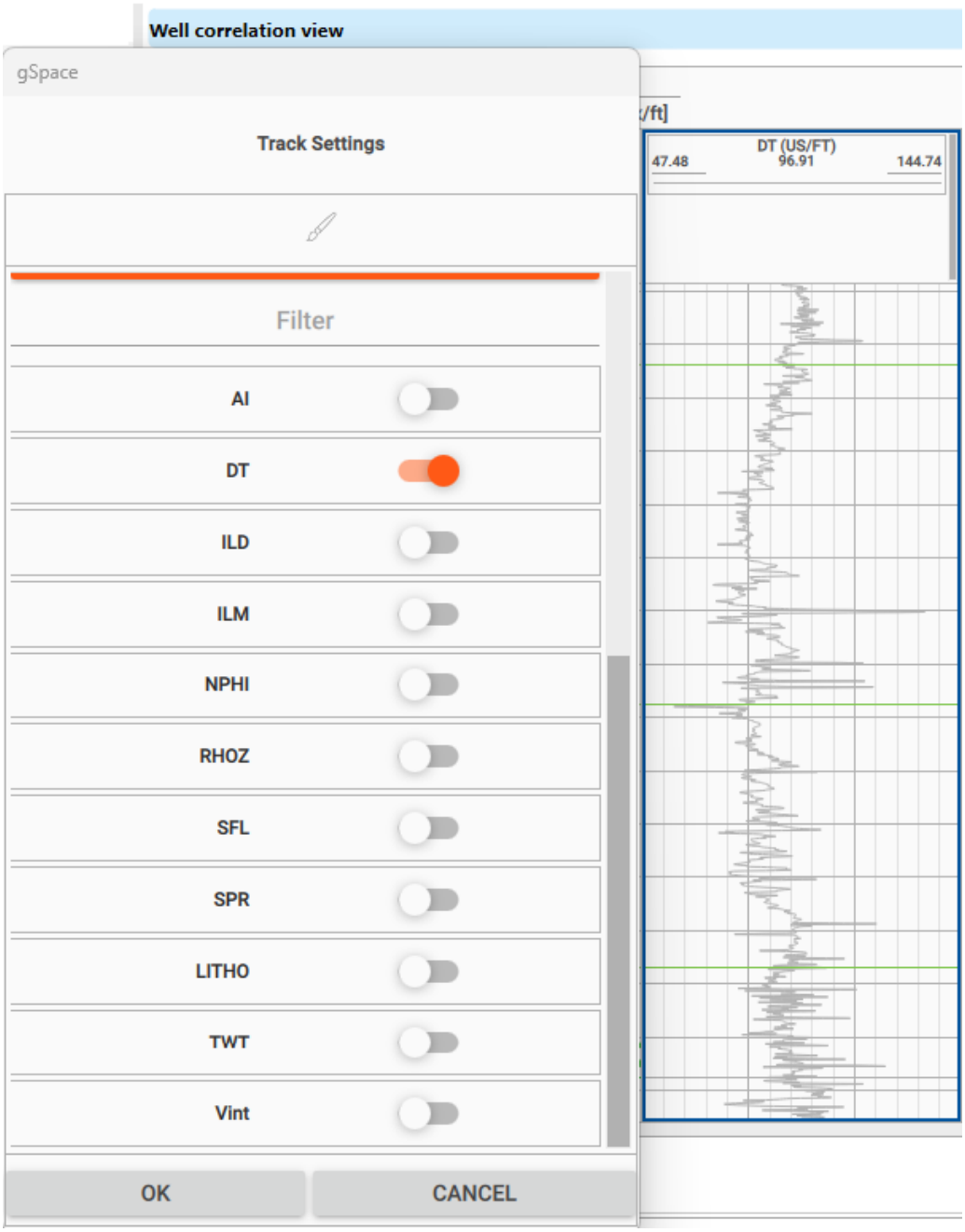
© 2026 Geomage 2003 LTD



Click on **UNUSED CURVES.** It will display all the available well logs. By default they all are turned OFF.

Select the desired well logs and they will automatically added to the new track as shown below. As soon as they are added, they will disappear from the UNUSED CURVES list.

© 2026 Geomage 2003 LTD



All these curves are added with default settings. To change the display settings of the individual curves, **Right click** inside the curve panel.

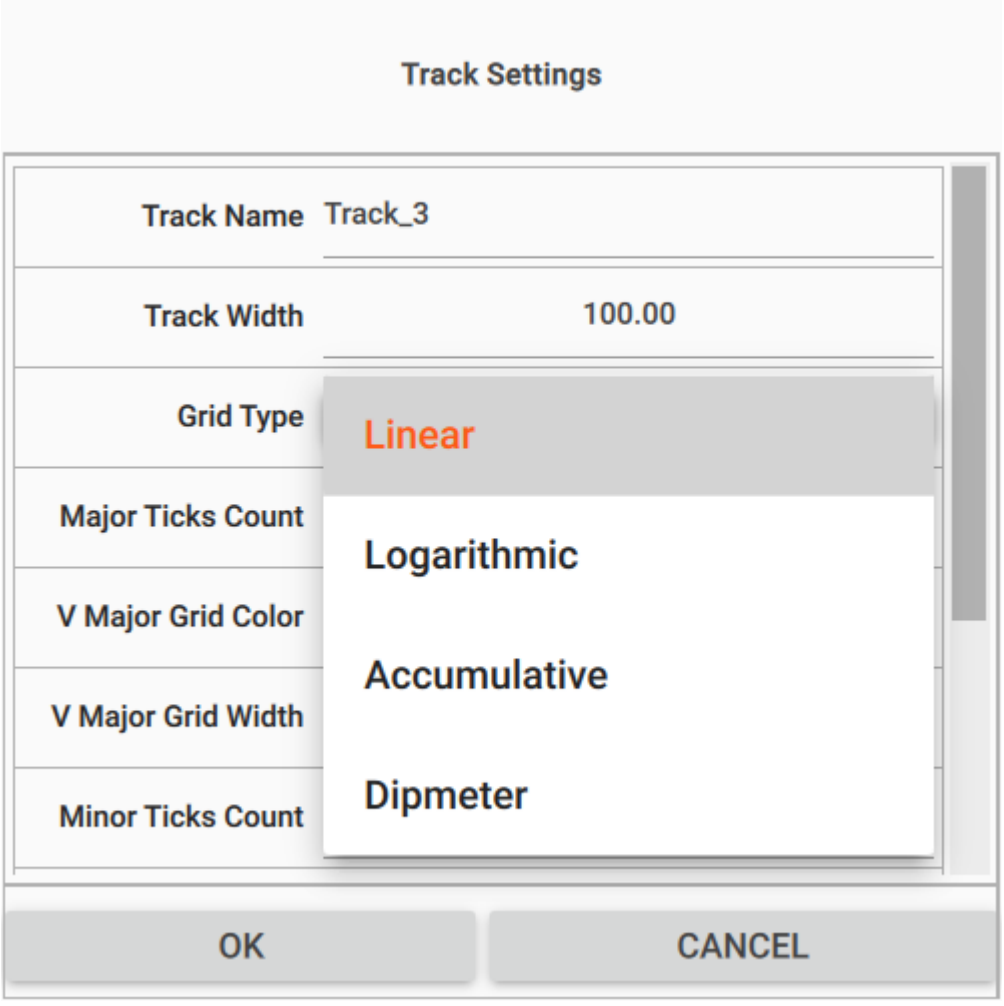
© 2026 Geomage 2003 LTD

Track Settings

Each track within the Well Correlation view can be individually configured for optimal visualization of log data.

To access the **Track Settings**, right-click within the desired track area and select **Advanced Track Settings**.

A new settings window will appear, allowing the user to configure the track's properties.



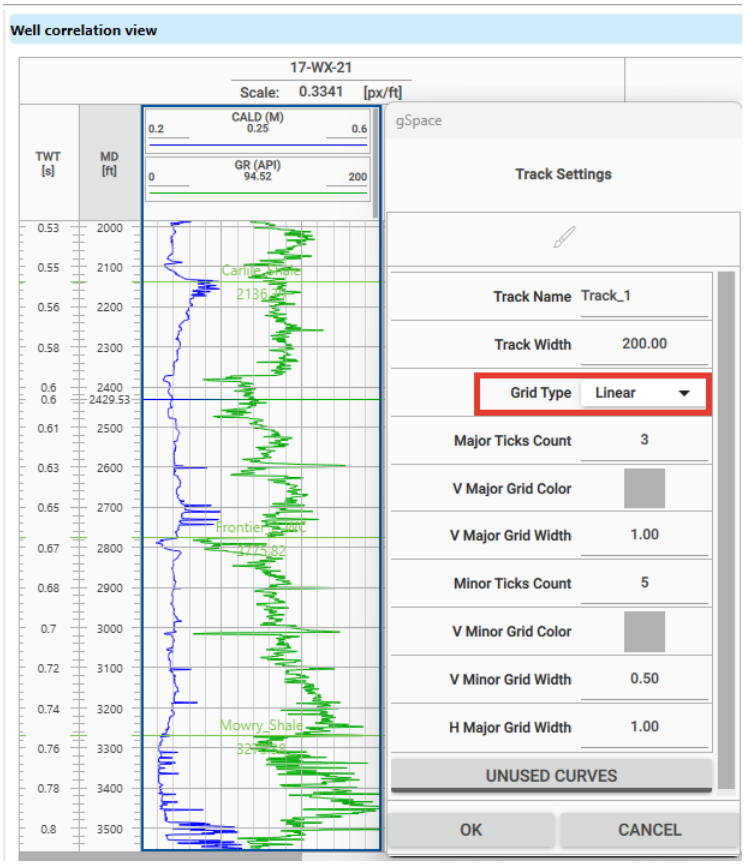
The following options are available:

- **Track Name**: Name of the track (e.g., GR, Density).
- **Track Width**: Controls the visual width of the track in pixels. A new track starts at 200 pixels, and a track that receives a seismic gather starts wider. The same width can also be

© 2026 Geomage 2003 LTD

set directly in the view: press the left mouse button on the right border of the track and drag horizontally to resize it while the button is held, down to a minimum of 50 pixels. An open **Track Settings** window follows the drag, and the resulting width is kept with the view like the other track settings.

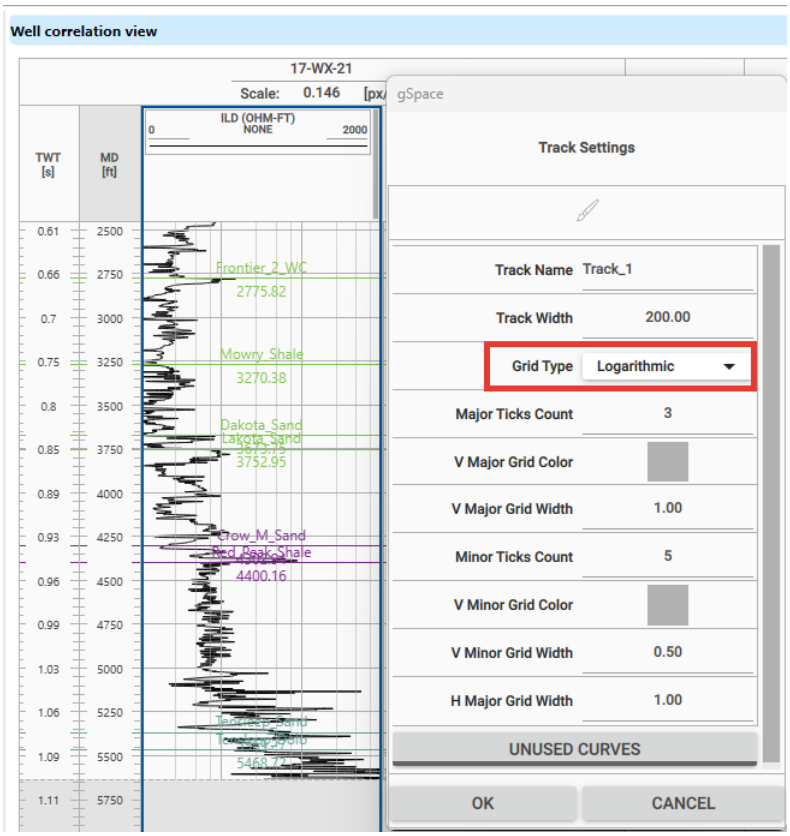
- **Grid Type**: Defines how the values in the track are scaled and drawn. The options include:
  - **Linear** *(default)*: Standard linear scale, most commonly used for continuous log data.



  - **Logarithmic**: Useful for logs with values that span several orders of magnitude, such as **Resistivity** (ILD, LLD).

This type of curve benefits from logarithmic scaling to better distinguish low and high values in the same track.

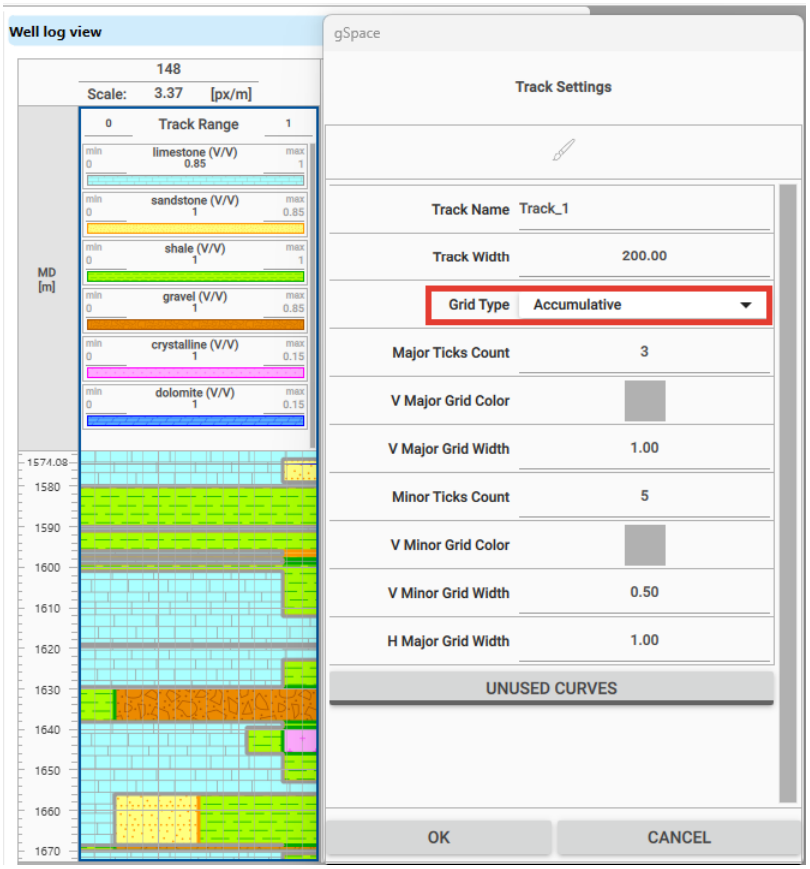
© 2026 Geomage 2003 LTD



- **Accumulative**: Ideal for plotting **cumulative values**, such as **net pay**, **cumulative porosity**, or **total thickness** curves.

This setting allows the user to display running totals, which are especially useful in reservoir analysis and zone summation.

© 2026 Geomage 2003 LTD



- **Dipmeter**: Draws structural dip data, such as bedding, fractures, faults and joints, as a tadpole plot instead of curves.

Each sample is drawn as a filled circle with a straight tail. The horizontal position of the circle is the dip angle on the dip scale of the track, and the tail points in the compass direction of the dip azimuth on the page, where 0° points up (north) and 90° points to the right (east), regardless of whether depth grows downwards. Tadpoles are drawn only at the depths that have valid values in both required curves; the data is not resampled.

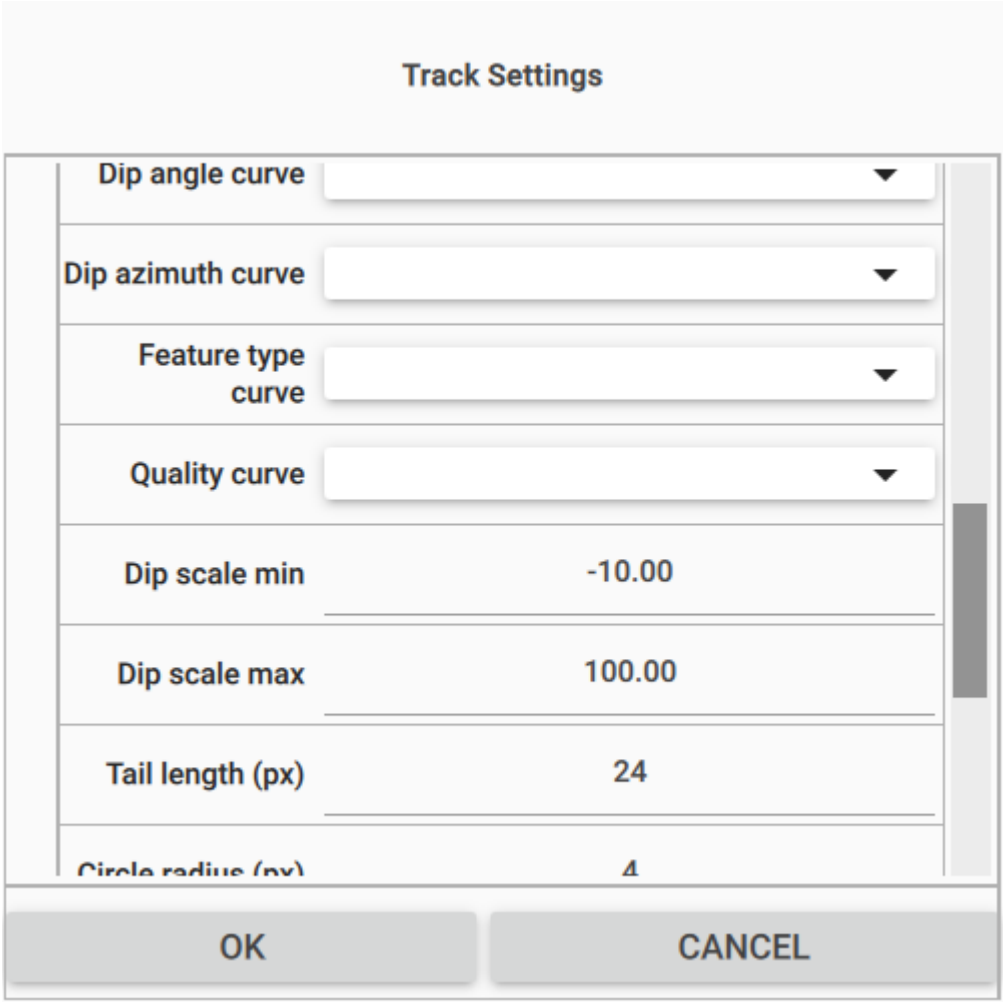
The track header shows **Dipmeter** followed by the current dip scale in parentheses, for example **Dipmeter (-10–100°)**. Hovering the cursor over a depth shows **Depth**, **Dip** and **Azimuth**, and also **Feature** and **Quality** when those optional curves are bound.

When **Grid Type** is set to **Dipmeter**, the **Track Settings** window gains a **Dipmeter** group with the following options:

- **Dip angle curve**: The curve that supplies the dip angle. Required.
- **Dip azimuth curve**: The curve that supplies the dip azimuth. Required.
- **Feature type curve**: Optional curve that assigns each sample to a feature type, which sets the color of its circle.
- **Quality curve**: Optional curve whose value is shown when the cursor is hovered over a sample.

© 2026 Geomage 2003 LTD

- **Dip scale min** and **Dip scale max**: The dip range that the width of the track covers, -10 and 100 by default.
- **Tail length (px)**: Length of the tadpole tail in pixels, 24 by default.
- **Circle radius (px)**: Radius of the tadpole circle in pixels, 4 by default.
- One color swatch per feature type: **Bedding** (blue), **Fracture** (red), **Fault** (black), **Joint** (green), **Unconformity** (magenta) and **Other** (gray). When no feature type curve is bound, all circles are drawn in gray.



Dip data is prepared as ordinary log curves and loaded with the standard LAS, ASCII and CSV import. The import recognizes the common dip mnemonics: DIPA, DIP_ANGLE and DIPDEG as Dip Angle; DAZI, AZIM, DIPAZ, DIP_AZIMUTH and DAZIM as Dip Azimuth; DIPTYP and DIP_TYPE as Dip Feature Type; and DIPQ, DIP_QUAL and DIPQUAL as Dip Quality, so the curves can be selected directly in the lists above. Each well in a well section has its own Dipmeter track, and the settings are stored with the project like the other track settings.

- **Major Ticks Count**: Number of major vertical grid divisions.

© 2026 Geomage 2003 LTD

- **V Major Grid Color / Width**: Set the color and width for major vertical grid lines.
- **Minor Ticks Count**: Number of minor divisions between major grid lines.
- **V Minor Grid Color / Width**: Set the color and width for minor vertical grid lines.
- **H Major Grid Width**: Adjusts the thickness of horizontal major grid lines.

Curve Settings

To customize how individual well logs are displayed within a track, the user can right-click on the log curve and select **Curve Settings**.
This opens a window where various visual and functional options can be adjusted. The example below shows the settings of a Filled Lines curve that uses a palette.

© 2026 Geomage 2003 LTD

Key Options in Curve Settings:

- **Curve Nickname**: A user-friendly label displayed on the track (e.g., "GR" instead of "Gamma Ray").
- **Units**: Measurement units for the curve (e.g., "API", "Ω·m", "g/cc").
- **Precision**: Number of decimal places shown for the curve values.

© 2026 Geomage 2003 LTD

- **Curve Type**: Defines the visual representation of the curve. Available options:
  - **Points**: Displays individual log values as discrete points.
  - **Lines**: Connects data points with smooth lines. Suitable for most continuous log curves.
  - **Filled Lines**: Similar to Lines, but fills the area under the curve—commonly used for porosity or saturation to visualize zone fill.
  - **Lithology**: Displays lithology logs with symbolic or color-filled styles.
  - **Spikes**: Draws each sample as a vertical spike from the baseline, useful for sparse or event-style logs.

Notes

- *Lines and Filled Lines are best for continuous logs (e.g., GR, DT, Density).*
- *Points may suit sparse or point data (e.g., deviation surveys).*
- *Lithology is for categorical data (e.g., rock type).*
- *Spikes is useful for sparse or event-style logs; for scanned raster logs use the Raster logs feature instead.*

- **Step Style** (Available only for "Lines" and "Filled Lines"): Defines how curve segments transition between depth values:
  - **No Stepping**: Default; lines are drawn directly between data points.
  - **Step Up**: Horizontal line at the top of each interval, then vertical drop.
  - **Step Center**: Horizontal line transitions midway between values.
  - **Step Down**: Horizontal line at the bottom of the interval, then vertical rise.

Notes

Step styles are helpful when displaying categorical or blocky data, such as lithology, saturation zones, or interpreted layers.

For example, Step Down might be used to show interval top values, while Step Up could reflect base markers.

- **Middle Point Fill** (Available only for "Filled Lines" curves on tracks whose Grid Type is not "Accumulative"): Controls where the fill of a Filled Lines curve starts. By default this option is off and the area is filled relative to the curve itself (its left or right side). When enabled, the fill is drawn relative to a fixed reference line instead of the curve edge.
- **Middle Value**: Appears when Middle Point Fill is enabled and Use Palette is off; if the curve uses a palette, this row is hidden. Sets the value of the reference (middle) line; samples with values on either side of this line are filled with the corresponding left or right side pattern.
- **Visible**: Toggle to show or hide the curve.

© 2026 Geomage 2003 LTD

- **Curve Width**: Adjusts the thickness of the curve line.
- **Transparency Level**: Set the curve's transparency (useful for overlapping curves).
- **Use Antialiasing**: Enables smoother curve edges for better visual quality.
- **Curve Color**: Manually set the color used to display the curve.
- **Propagate Curve Data**: Copies the current curve display settings to the same curve in the other wells of the section.

To remove any of these curves, the user can simply click on the Delete Curve icon. By default it is Grey in color. As the user hover the cursor, it will turn into Red.

Working with Palettes

Instead of drawing a curve in a single color, the user can color it by value using a palette. In **Curve Settings**, enable **Use Palette** and then choose a predefined palette (colormap) from the **Palette** selector. Each sample is then shaded according to its value. See Working with Palettes 323 for details.

The palette editor lets the user adjust the color range and add or move color stops. The edited palette can be kept **Local** to this curve only, or switched to **Global** so it is saved and can be reused for other curves.

Note: Use Palette applies to curves drawn as Filled Lines.

For lithology curves, the curve menu provides a **Lithology Map** action, which lets the user assign a color and fill pattern to each lithology class.

Raster logs

Well log raster images (for example scanned strip logs) can be displayed in the Well Correlation view alongside curves, markers and comments. Rasters are imported per well using the Open raster 142 wizard.

Depth maps

A depth map that is ticked under **Maps** in the Data Manager 124 is also displayed in the Well Correlation view, as a horizontal line that crosses every well. The line is drawn at the measured depth at which the well trajectory actually meets the map surface, so deviated wells are placed correctly instead of at the plain true vertical depth of the map.

A well that does not cross the area of the map, or that has no valid trajectory data, simply shows no line for that map. Clearing the checkbox of the map in the Data Manager hides its lines again.

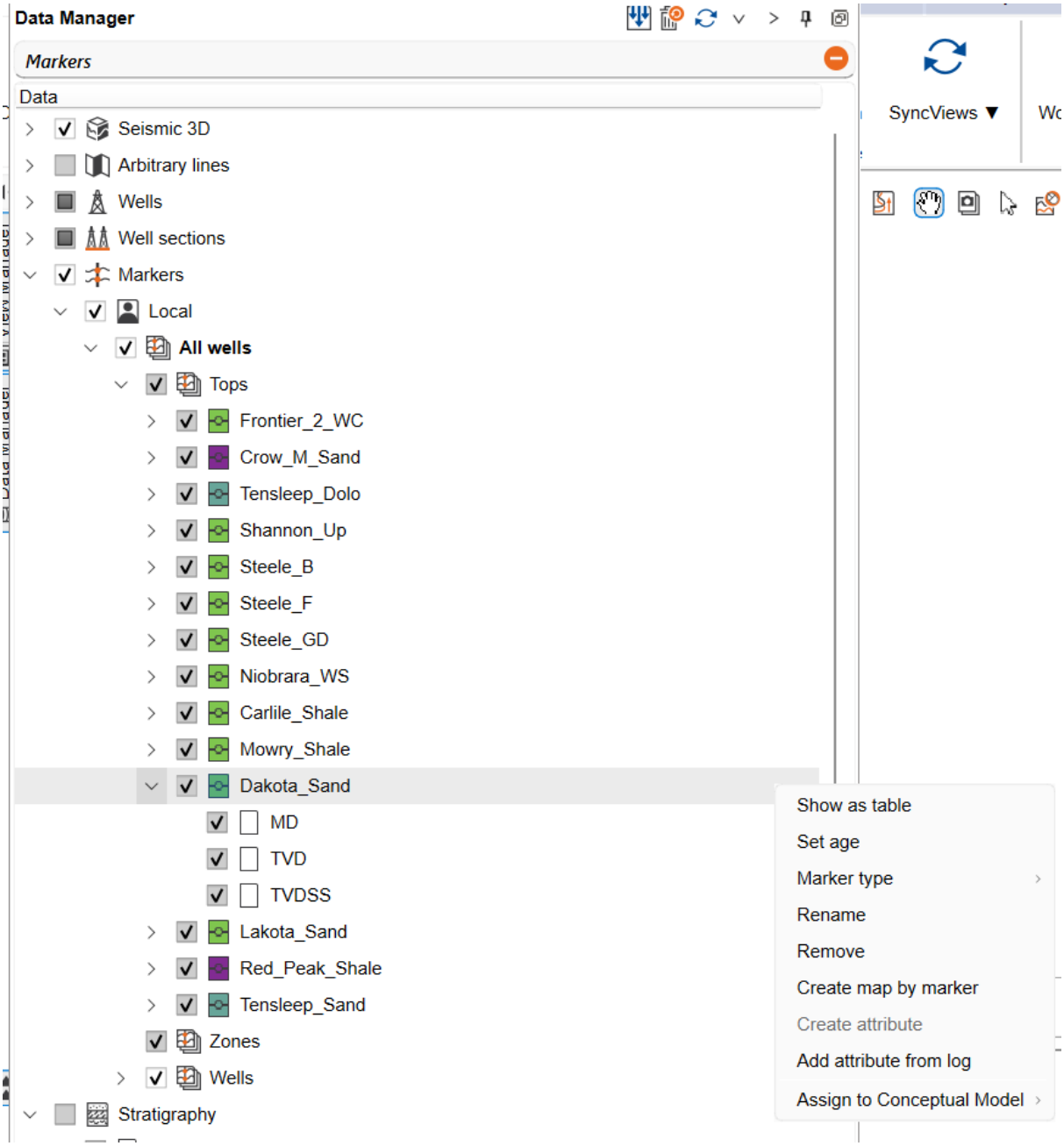
Every displayed map is given an automatic default color. To change how its line is drawn, open **Global Settings** and use the **Map:** group of that map, described above.

A map line is read-only in the Well Correlation view: it cannot be dragged to another depth, and moving the cursor over it keeps the ordinary pointer instead of the double arrow that

© 2026 Geomage 2003 LTD

appears on a marker line. Its depth always follows the stored map, so the line stays consistent with the map value at each well; to change a map, use the Map editor 341 view. Right-clicking a map line still opens its display settings, and well markers keep their usual editing behavior, described in Working with Markers 551.

Managing Marker Sets in the Data Manager



© 2026 Geomage 2003 LTD

The Data Manager interface allows users to manage and customize marker sets for wells. The following functionalities are available:

1. **Marker Set Options (Left Panel):**
    - **Set Age:** Assign a specific geological age to the selected marker set.
    - **Marker Type:** Classify markers as either Horizon, Fault, or Intrusion. This helps in categorizing and visualizing different types of geological features.
    - **Rename:** Change the name of the selected marker set.
    - **Remove:** Delete the selected marker set from the project.
    - **Create Map by Marker:** Generate a map based on the selected marker, visualizing its distribution across the study area.

2. **New Marker Set Management (Bottom Left Panel):**

    - This panel allows the creation of a new marker set. Once created, the new marker set appears in the list, ready for further customization and data input.

3. **Marker Set Actions (Right Panel):**
    - **Set Default:** Designate the selected marker set as the default for the project.
    - **Rename:** Change the name of the selected marker set.
    - **Remove:** Delete the selected marker set.
    - **Duplicate:** Create a copy of the selected marker set, useful for making modifications without altering the original.
    - **Export/Import Markers Struct:** Export or import the structure of the markers for use in other projects or sessions.
    - **Export/Import Markers Ages**: Export or import the ages of the markers, ensuring consistency in geological timelines.
    - **Sort by Depth/Age:** Organize markers within the set either by their depth in the well or by their geological age.
    - **Commit:** Save changes made to the marker set.
    - **Set as Master:** Designate the selected marker set as the primary reference for correlation and mapping.
    - **Show Revisions:** View the history of changes made to the marker set.
    - **Add Group/Unconformity:** Add a new group or unconformity marker to the set, useful for defining significant geological boundaries.
    - **Add Marker:** Insert a new marker into the set.
    - **Child Build Order:** Adjust the build order of child markers within the set, either From top to down or From down to top. This controls the sequence in which markers are applied or visualized.

© 2026 Geomage 2003 LTD

3.4.7.1 LAS editing

The **Edit LAS curve** tool applies processing and editing operations to individual log curves.

This is useful for cleaning, correcting, or customizing log data before interpretation or correlation.

To open the wizard, go to the **Petrophysics** ribbon tab and, in the **Logs** group, click **Edit LAS curve**.

You can also open the wizard for a particular curve from the Well correlation view 296: activate the **Select interval** tool on the view header toolbar and select the curve in its track. The wizard opens with that well and curve already chosen. To preset the interval as well, drag along the curve with the same tool: the wizard then enables **Start** and **End** on the **MD range** tab and fills them with the depths you covered. In a track that holds several curves, select the curve before you drag; otherwise the view reports *Please, select a curve*.

This route works only with the **Select interval** tool. While any other tool of the view header toolbar is active, clicking or dragging a curve leaves the wizard closed, so switch tools first or open the wizard from the **Petrophysics** ribbon tab.

The wizard consists of the following key areas:

1. Well and Curve Selection

At the top, select the target **Well** and the **Curve** (log) you wish to edit using the dropdown menus.

Enable **Batch calculation** to apply the same editing action to more than one well in a single run. The adjacent dropdown sets the scope of wells to process: besides **All wells**, it also lists the project's **Filters**, **Folders** and **Well sections**, so you can restrict the operation to a saved filter, a well folder or a well-section group. The **Well** and **Curve** dropdowns still choose which curve is edited in each well.

2. Depth Range

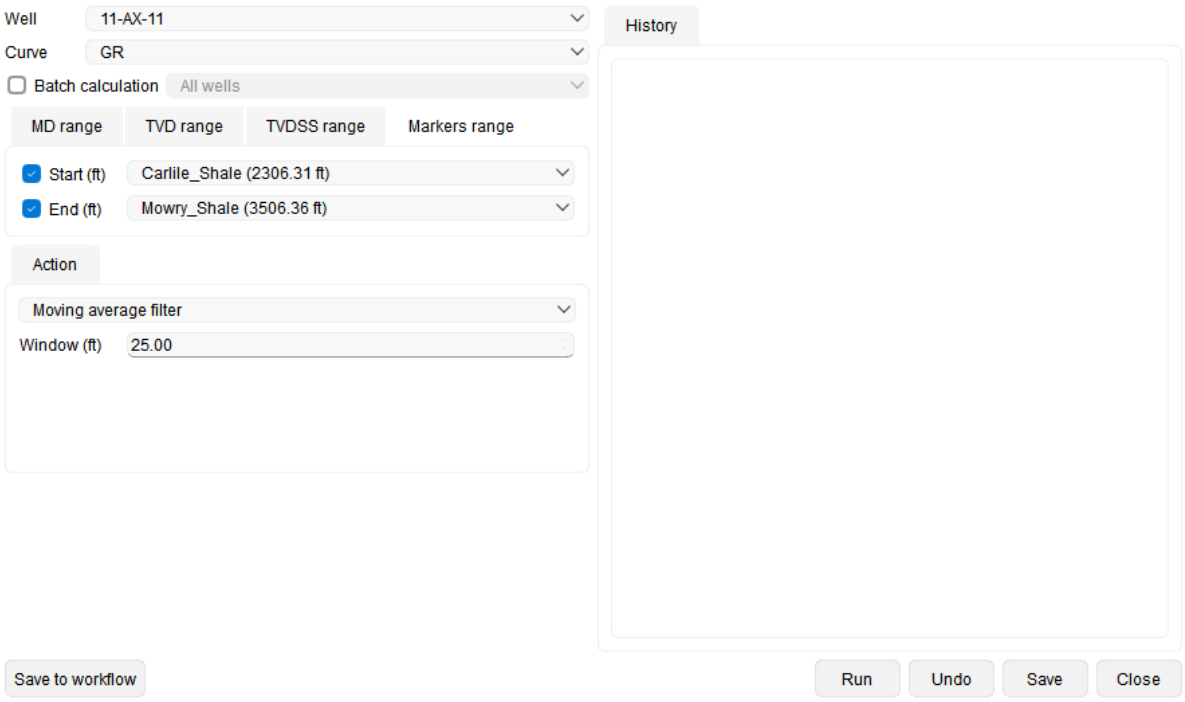
Select the depth range over which the operation will be applied. Options include:

- **MD range**: Measured Depth
- **TVD range**: True Vertical Depth
- **TVDSS range**: the vertical depth measured below mean sea level
- **Markers range**: Use tops/markers to define interval

© 2026 Geomage 2003 LTD

On the **MD range**, **TVD range**, or **TVDSS range** tab, enable **Start** and/or **End** and enter the limiting depth values.

On the **Markers range** tab, enable **Start** and/or **End** and choose the corresponding marker. The example below applies **Moving average filter** between two selected markers.



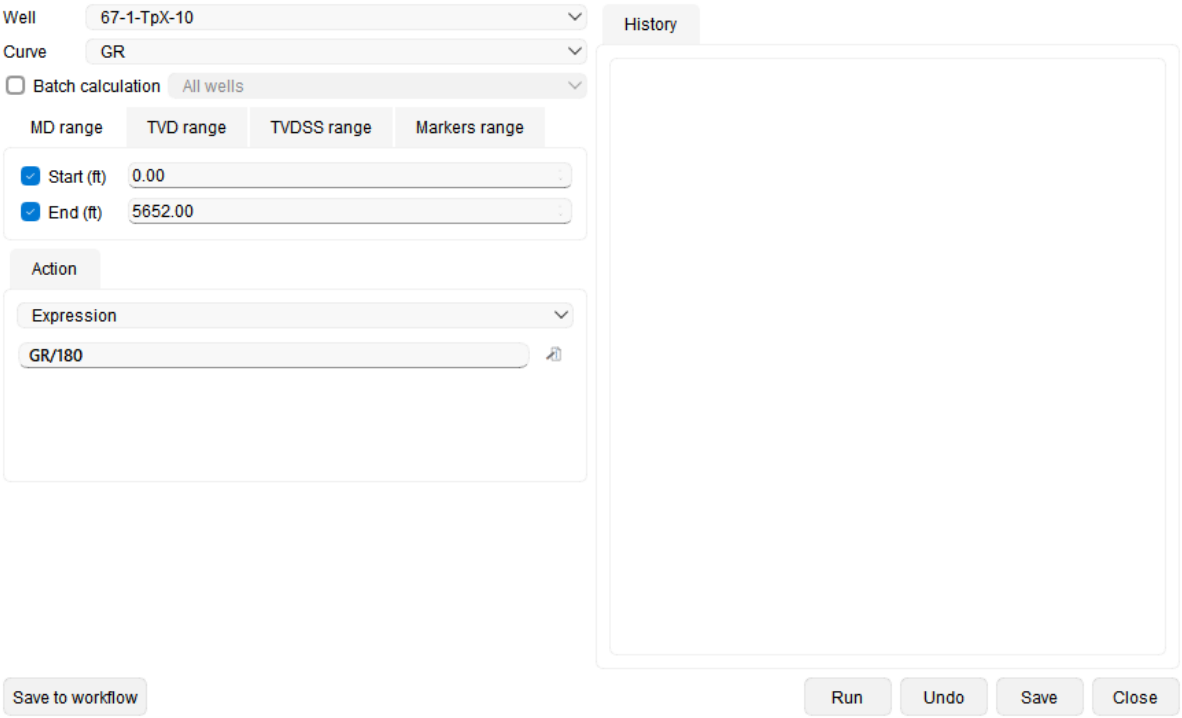
3. Action – Log Editing Options

In this section, choose one of the available operations from the **Action** dropdown menu:

- **Moving average filter**: Smooths the log by averaging values within a specified window. Enter the **window size** in the current depth unit. The field is labelled **Window (m)** or **Window (ft)**, depending on the measurement system.
- **Median filter**: Reduces noise by replacing each value with the median of its surrounding values within the defined window.
- **Bandpass filter**: Applies frequency-based filtering to isolate or remove specific ranges in the data.
- **Depth shift**: Shifts the entire curve vertically by a fixed amount. Useful for aligning logs when depth mismatches occur (e.g., from tool calibration errors).
- **Expression**: Calculates new values for the selected curve from an expression. Enter the formula directly or click the calculator icon to build it from the available curves, operators and functions.

© 2026 Geomage 2003 LTD

The **Expression** action edits the selected curve in place. Enter only the expression, for example **GR/180**; do not add a **name =** prefix. If you include a prefix in the **Log calculator** opened from the calculator icon, g-Space asks you to remove it. You can reference the selected curve by its displayed name or by **value**. Curve names such as **SP** and **SP1** are treated as separate identifiers.



4. History Panel

The **History** tab lists the actions run during the current editing session. Use **Undo** to remove the most recent action.

5. Buttons

- **Save to workflow**: Adds the configured edit as a workflow task.
- **Run**: Applies the selected action and adds it to History while keeping the wizard open.
- **Undo**: Removes the most recently run action and restores the previous edit state.
- **Save**: Saves the accumulated edits and closes the wizard. In the overwrite prompt, choose **Yes** to replace existing values, **No** to enter new curve names and keep the originals, or **Cancel** to return without saving. When you choose **No** and save new curve names, the new curves are added to the **Data Manager** and displayed immediately in the same **Well correlation view** track or tracks as their originals. They inherit the originals' display settings and use a contrasting color.

© 2026 Geomage 2003 LTD

- **Close**: Closes the wizard. If the History contains unsaved actions, confirm whether to discard them.

Deleting a single curve

To delete one log curve without affecting the rest of the LAS, right-click that curve in the Data Manager (in the well's log list or under **All curves**) and choose **Remove**. g-Space asks for confirmation (*Do you want to remove selected las curves?*) and then removes only the selected curve, leaving the other curves in the file unchanged.

3.4.7.2 Working with Palettes

Palettes control how well data is coloured in the *Well correlation view*. Instead of drawing a curve in a single fixed colour, a palette maps colour to value, which makes continuous logs and interpreted classes (lithology, facies, zones, saturation) easier to read and compare between wells.

Enabling a Palette

Right-click a curve header in the correlation view to open **Curve Settings**. Set **Curve Type** to *Filled Lines*, choose a **Line Style** (Solid, Dot, Dash, Dash dot, or Dash dot dot), then turn on **Use Palette** and choose a **Palette** to colour the curve by value.

Choosing and Editing a Palette

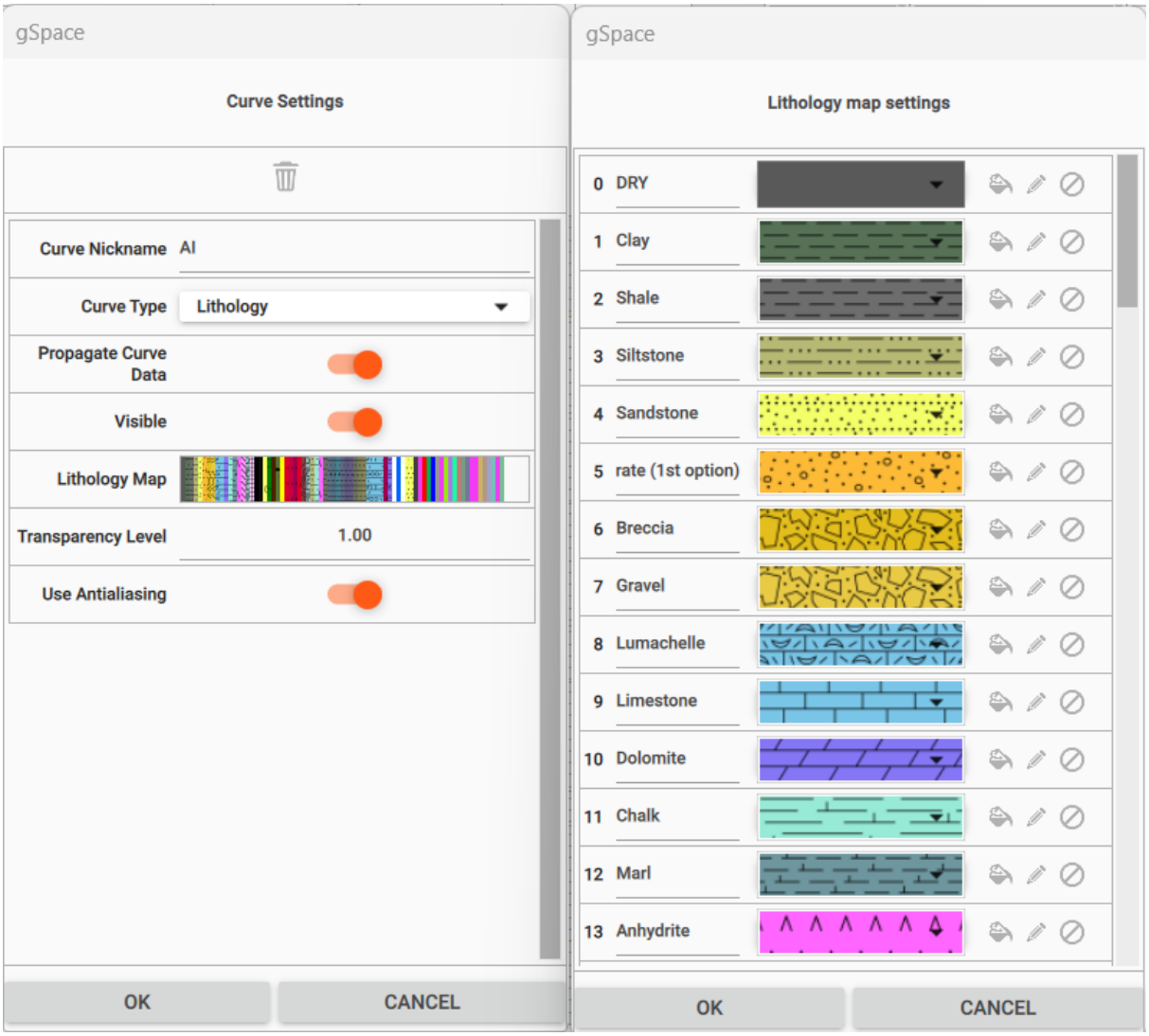
- **Select a palette** — choose a predefined colormap from the **Palette** selector in Curve Settings.
- **Edit colour ranges** — open the palette editor to adjust colour stops, value limits and the colour assigned to each interval.
- **Local or global** — keep the palette *Local* to the selected curve, or save it as *Global* to reuse it elsewhere in g-Space.
- **Check the value range** — confirm the colours match the expected curve range so key intervals stand out clearly.

© 2026 Geomage 2003 LTD

Lithology and Discrete Data

For discrete logs, set the **Curve Type** to **Lithology** in Curve Settings and open the **Lithology Map** to define how each class is drawn. Assign a colour and a fill pattern to every lithology class (for example clay, shale, sandstone, limestone or dolomite); use the colour picker to fine-tune class colours so the lithology column reads clearly across wells.

© 2026 Geomage 2003 LTD



Colormap Manager

The **Colormap Manager** is a project-wide tool for creating, duplicating, editing and deleting palettes. Open it from the **Tools** menu (**Colormap manager**). The palettes managed here are the same colormaps used elsewhere in g-Space, including the *Well correlation view*.

The dialog lists all palettes on the left and shows the selected palette's colour stops on the right. Use the buttons under the list to manage palettes:

- **New...** — create a new palette and give it a name. It starts as a simple black-to-white gradient that you can then edit.
- **Duplicate...** — copy the selected palette under a new name, producing an editable version of any palette.

© 2026 Geomage 2003 LTD

- **Delete** — remove the selected palette, after a confirmation prompt.

Selecting a palette shows its colour stops as a table of **Value** and **Color**. For an editable palette you can:

- Use **Add point** and **Remove point** to add or delete a colour stop.
- Edit a **Value** cell to change where a stop sits, and double-click a **Color** cell to pick its colour (transparency is supported).
- Click **Apply** to save the changes back to the palette. A palette must keep at least two colour points.

The standard palettes (**Stack**, **Velocity**, **Gray**, **HSV** and **Jet**) are read-only. To adapt one, select it and use **Duplicate...** to make an editable copy.

See Also

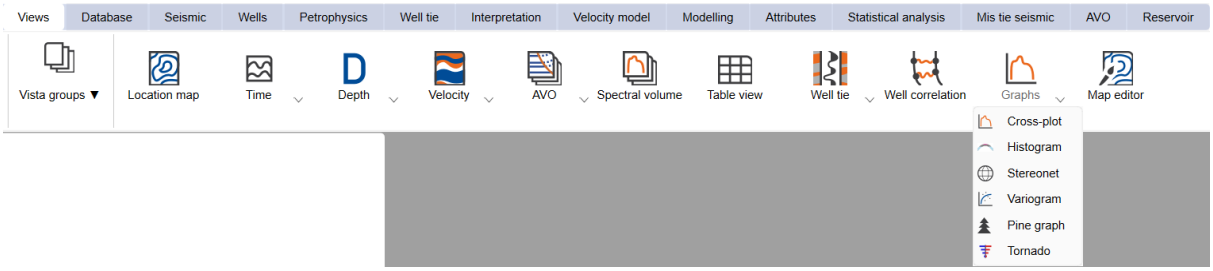
- Well correlation view 296
- Lithology 77
- Setting up a well template 544

3.4.8 Graphs

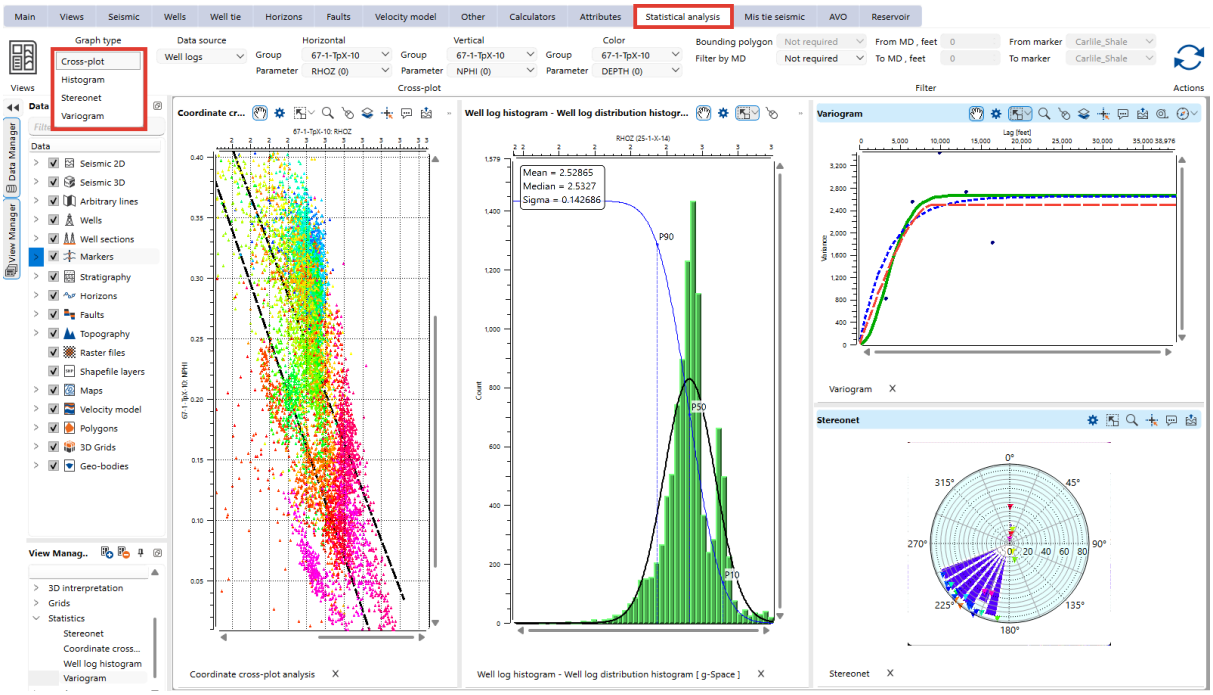
The **Graphs** section in the Views bar 38 provides tools for visualizing and analyzing relationships within your data. It includes the following graph types:

1. Cross-Plot 330: Visualize the relationship between two selected attributes plotted on the X and Y axes. This is particularly useful for identifying trends, correlations, and patterns in the data.
2. Histogram 327: Display the frequency distribution of well log data. Use histograms to analyze data ranges, identify dominant values, and detect outliers in well attributes.
3. Variogram 335: Visualizes spatial continuity by plotting semivariance against lag distance, aiding geostatistical analysis.
4. Stereonet 336: Plot structural orientations, such as dip and azimuth, on a stereographic projection. Stereonets are especially valuable for analyzing geological structures, such as faults, bedding planes, and stress orientations.
5. Pine graph 332: A quality-control tool for velocity models that plots the depth difference between well markers and the velocity-derived depth.
6. Tornado 338: Show how much the result of a formula changes when each of its input parameters varies between a minimum and a maximum value, which identifies the parameters that influence the result most.

© 2026 Geomage 2003 LTD



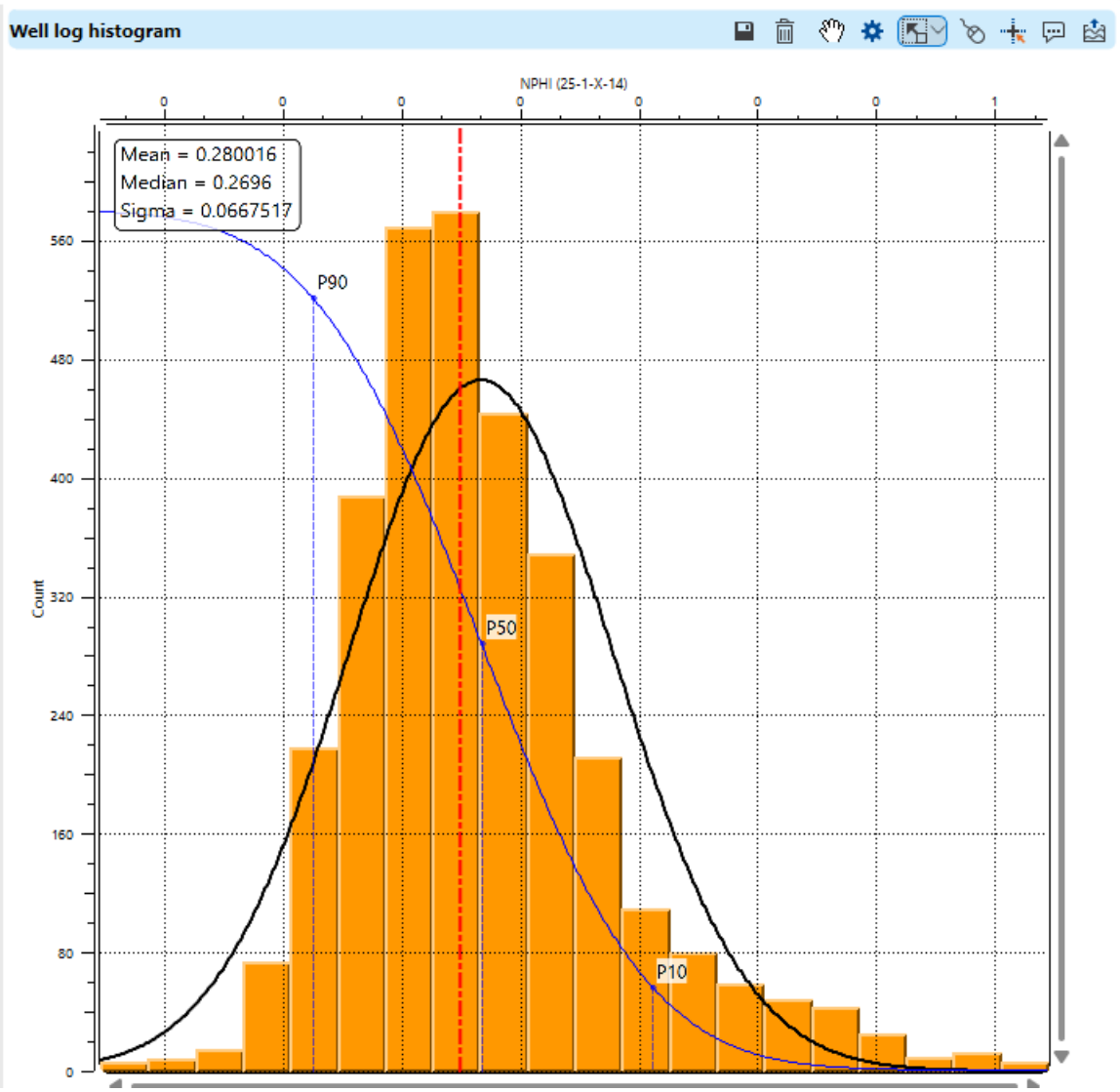
Configure the graph settings, such as data source, parameters, and filtering options in the [Statistical analysis bar](111).



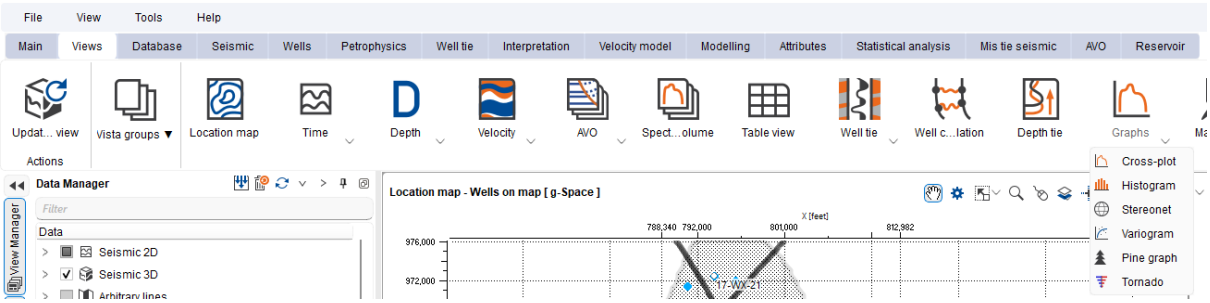
3.4.8.1 Histogram view

The Histogram view provides a graphical representation of the distribution of an attribute, giving users insights into the frequency and range of data values.

© 2026 Geomage 2003 LTD



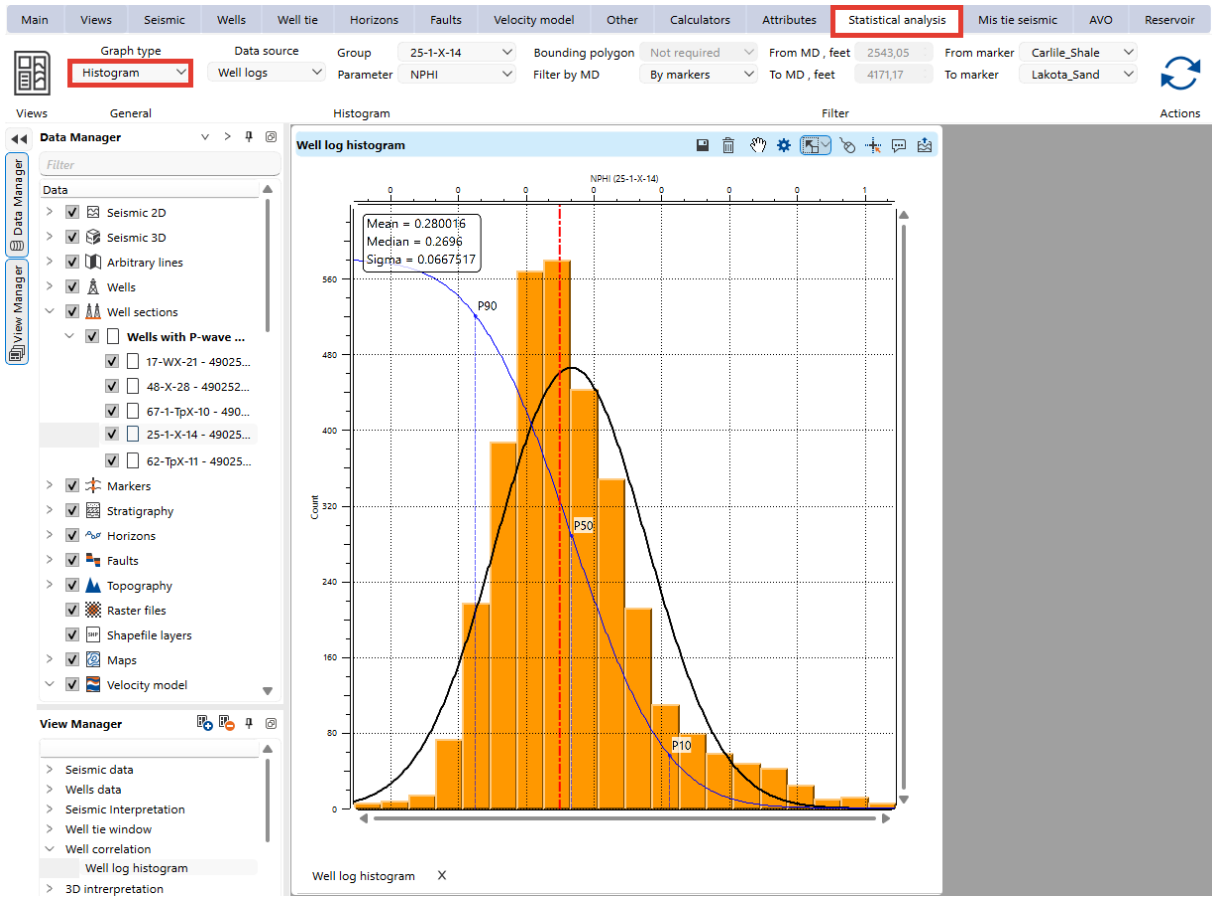
To create Histogram view go to the Views bar 38, navigate to **Graphs** and click on **Histogram** button:



© 2026 Geomage 2003 LTD

Each click on **Histogram** opens a new **Well log histogram** view. Every opened view keeps its own parameters, so the settings in the Statistical analysis bar apply to the active histogram view only.

To set what data to be analyzed set parameters in the Statistical analysis bar:



Functional panel allows user to perform the following actions:

- open Visual Settings panel
- adjust view area to all visible objects

© 2026 Geomage 2003 LTD

set control item

enable mouse tracker

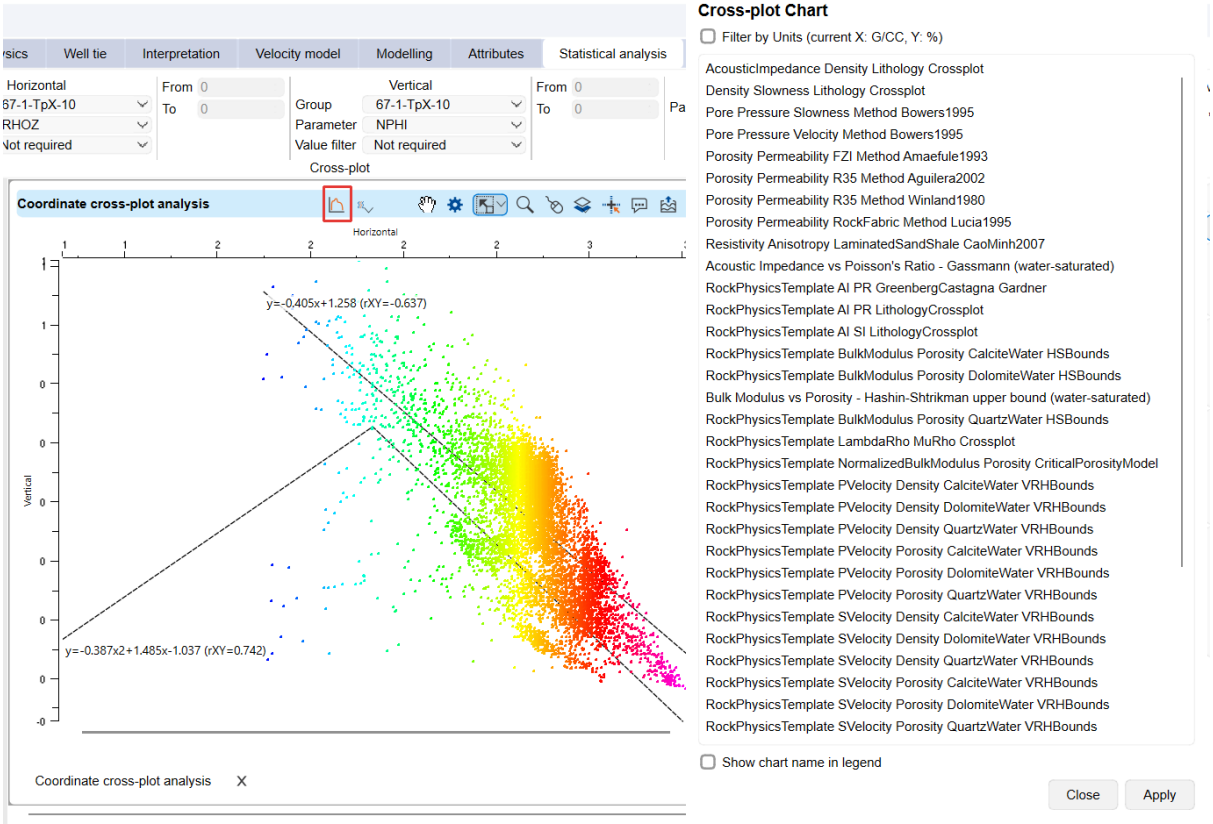
enable tool tip

export picture

For more information about the Histogram analysis please refer to Statistical analysis 742

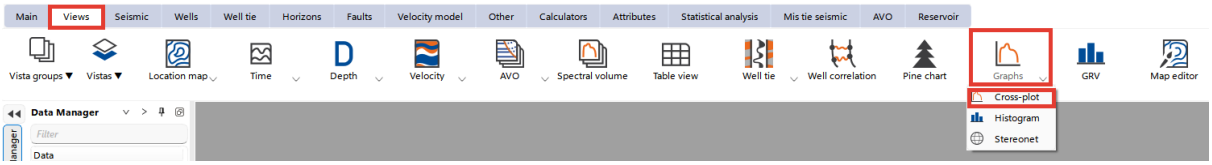
3.4.8.2 Cross plot view

Cross plot is a view which allows user to cross-plot different data types loaded in g-Space



To create cross plot view go to the Views bar 38, navigate to **Graphs** and click on **Cross plot** button:

© 2026 Geomage 2003 LTD



To set what data to be analyzed set parameters in the Statistical analysis 111 bar:

To define the object visibility stile go to the Visual Settings 206 by pressing in the view functional panel or find it in the panel 123 sets.

For more information about the cross-plotting please refer to Statistical analysis 742

Functional panel allows user to perform the following actions:

- open Visual Settings panel
- adjust view area to all visible objects
- adjust horizontal scale
- adjust vertical scale
- set/unset zoomable
- set control item
- add Object
- enable mouse tracker
- enable tool tip
- export picture
- ruler tool

© 2026 Geomage 2003 LTD

Chart library

The **Show Charts** button on the top toolbar of the Cross plot view opens the **Cross-plot Chart** library — a catalog of published reference charts that can be overlaid on the current cross-plot. The dialog lists the available charts and provides the following controls:

- **Filter by Units** — enabled by default. When on, only charts whose X and Y axis units match the currently selected Horizontal and Vertical parameters are listed. Turn it off to browse all available charts.
- Chart list — select a single chart to overlay it on the cross-plot. Hover over an entry to see its X and Y quantities, units, and source reference.
- **Show chart name in legend** — adds the selected chart's name to the plot legend.

If you select a chart whose units differ from the current axes, the dialog displays the warning *Selected chart units do not match current X-Y units.* in red. Selecting a chart updates the overlay immediately; toggling **Show chart name in legend** updates the legend immediately. Click **Apply** to reapply the current settings, or **Close** to dismiss the dialog.

Selection and polygon tools

The top toolbar of the Cross plot view provides interactive tools for isolating and annotating clusters of data points.

Selection tools

The selection tools let you enclose a cluster of points so that the regression is fitted to that subset only. Three shapes are available:

- **Selection: rectangle**
- **Selection: circle**
- **Selection: free-form**

When a selection shape is applied, points inside it become active and are used for the linear and quadratic regression calculation, while points outside appear pale and remain inactive. Only one selection shape can be active at a time. The shape is interactive — it can be moved or resized by dragging its handles. Click the active tool again to switch it off and clear the selection.

Draw polygons

The **Draw polygons** tool lets you create multiple named, coloured polygons for grouping and annotation. Unlike the selection tools, drawn polygons do not change the regression fit. Each polygon can be moved or reshaped, and displays a label with its name, the number of points it contains (N), and the average X and Y values of those points.

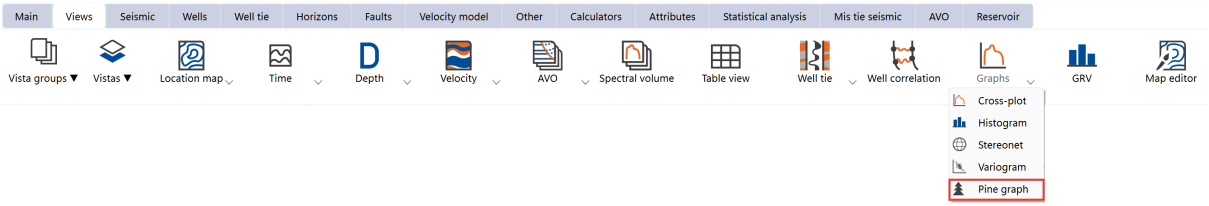
Right-click a drawn polygon to **Rename polygon...**, **Change colour...** or **Delete polygon**.

3.4.8.3 Pine graph view

Pine chart view is a quality control tool for velocity models. The Pine Chart View is a quality control tool designed specifically for evaluating and validating velocity models in geophysical analysis. This view provides a visual representation of velocity data across various depths or intervals, allowing users to assess the consistency and accuracy of the velocity model.

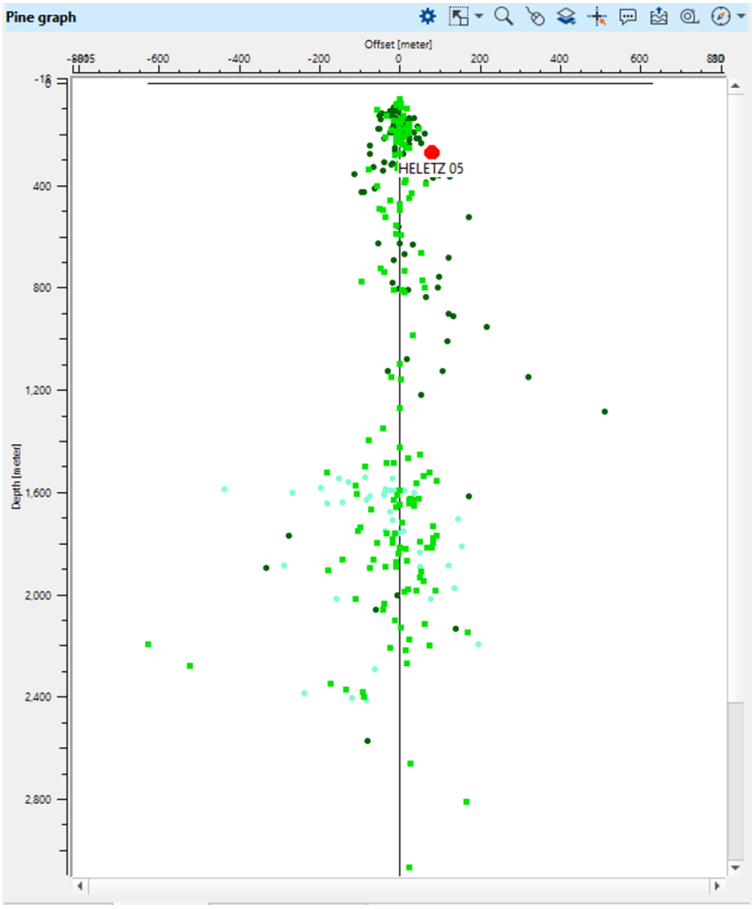
© 2026 Geomage 2003 LTD

To create pine chart view go to the Views bar 38 in the Ribbon bar 33 and click on **Pine Chart** option:



The newly created view will appear in the View manager 189 in the active Workspase 224. User can place Pine graph view by dragging it to any workspace except pre-defined once (Seismic data 228 and Well data 232).

Each point on this graph is a well marker. The x-axis shows the difference between velocity based depth and the actual marker depth. The y-axis shows the marker depth.



Right-click the graph to store it in the project: **Save Pine graph to Data Manager...** saves the graph as an object and **Save Pine graph picture to Data Manager...** saves a static picture of

© 2026 Geomage 2003 LTD

the current view. Both commands ask for a **Name** and place the new item in the **Pine graph** folder of the **Statistics** category of the Data manager 124. Saving the graph as an object requires an active velocity model: the saved object refers to that model, and **Open** makes it current again and redraws the graph. See Statistical analysis 742 for the commands shared by all saved graphs.

To define the object visibility stile go to the Visual Settings 206 by pressing in the view functional panel or find it in the panel 123 sets.

Functional panel allows user to perform the following actions:

open Visual Settings panel

adjust view area to all visible objects

 adjust horizontal scale

adjust vertical scale

set/unset zoomable

set control item

add Object

enable mouse tracker

enable tool tip

export picture

ruler tool

show compass

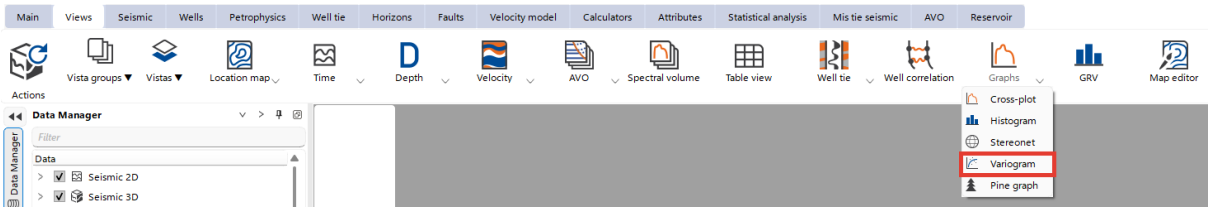
show minimap

© 2026 Geomage 2003 LTD

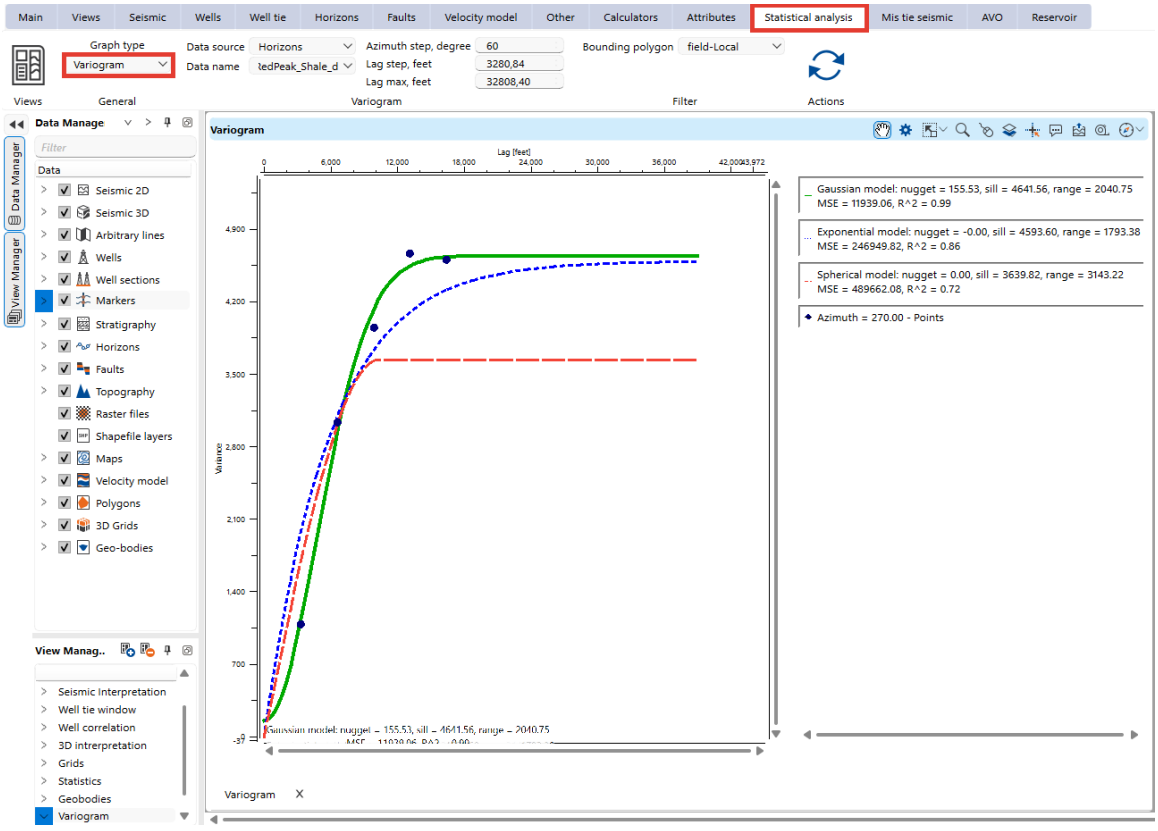
show scale bar

3.4.8.4 Variogram view

To create Variogram view go to the Views bar 38, navigate to **Graphs** and click on **Variogram** button:



To start variogram analysis, navigate to to the Statistical Analysis 742 bar and select the **Variogram** option under the Graph type dropdown menu.



In the Visual Settings panel, users can adjust visual settings for better interpretation:

- Transparency levels

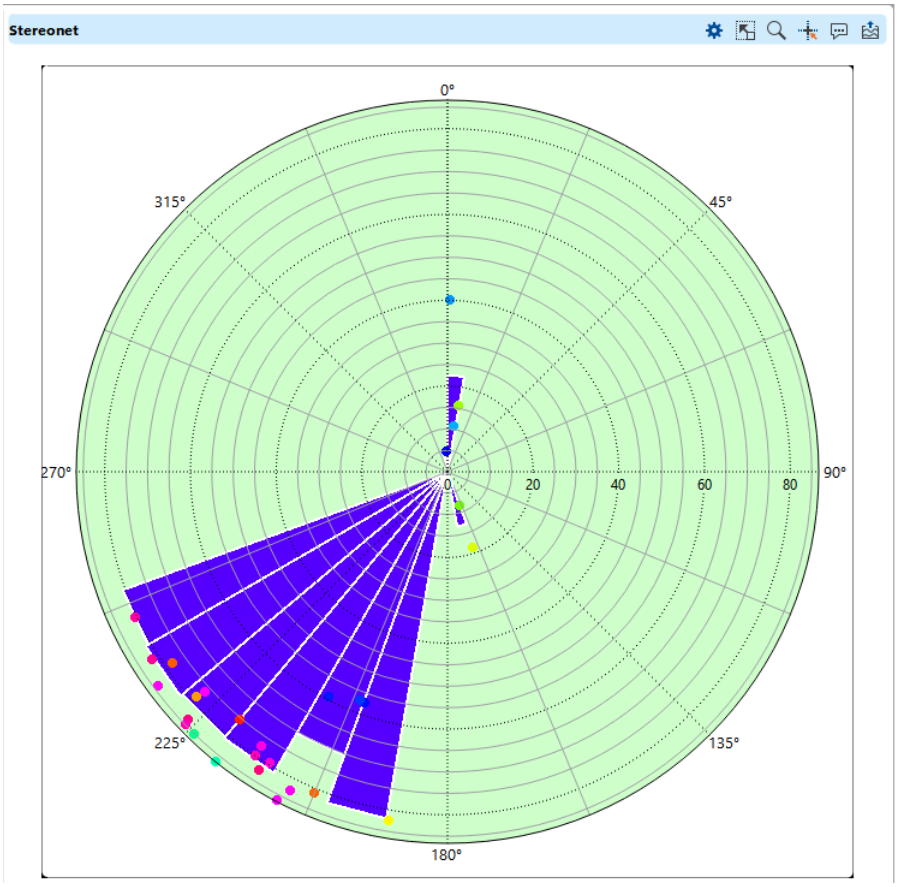
© 2026 Geomage 2003 LTD

- Color schemes
- Line types and thickness
- Point size and shape

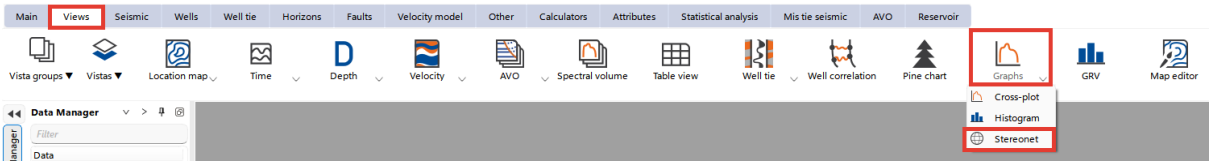
Users can also enable or disable specific models to focus on relevant data trends.

3.4.8.5 Stereonet view

The Stereonet View is a view that allows users to visualize and analyze orientation data, such as azimuths and dips, in g-Space.

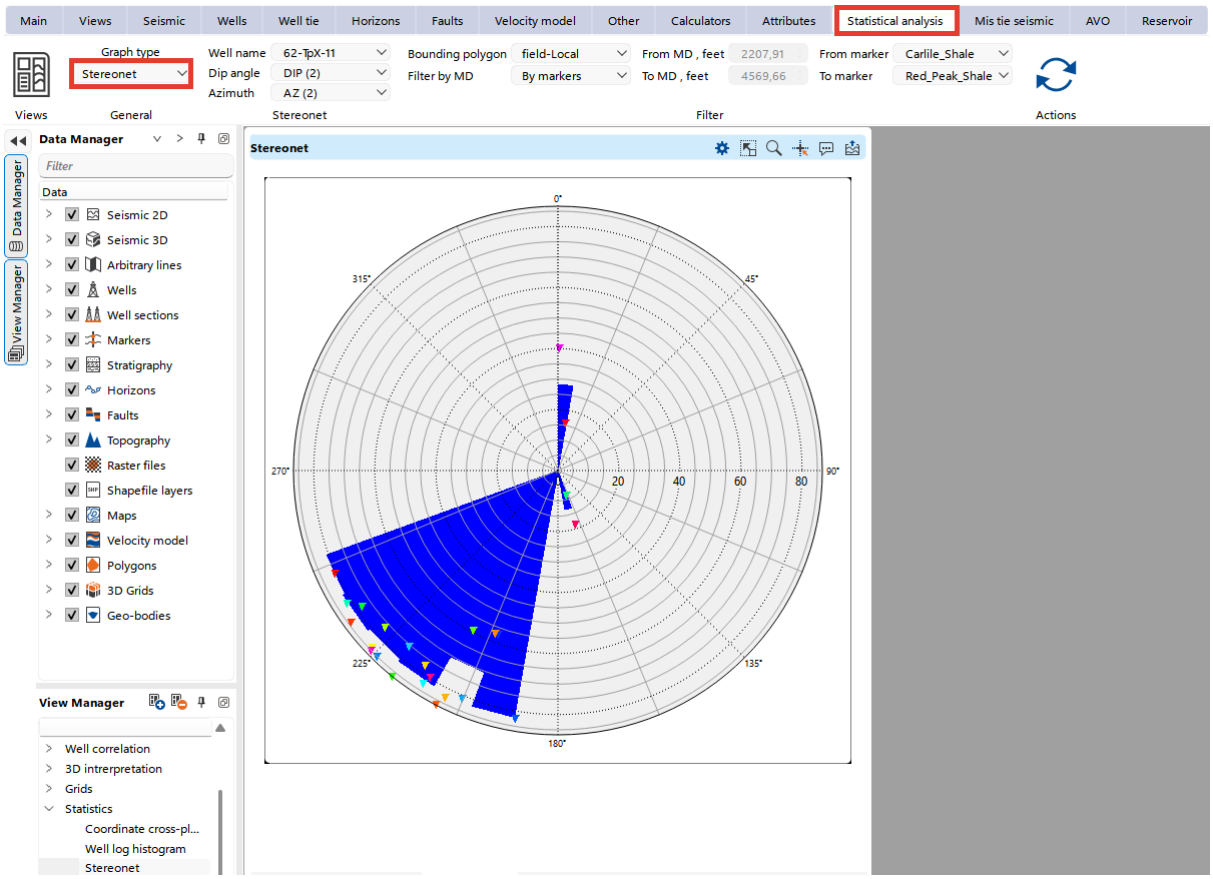


To create Stereonet view go to the Views bar 38, navigate to **Graphs** and click on **Stereonet** button:



© 2026 Geomage 2003 LTD

To set what data to be analyzed set parameters in the Statistical analysis bar:



Functional panel allows user to perform the following actions:

- open Visual Settings panel
- adjust view area to all visible objects
- set/unset zoomable
- enable mouse tracker
- enable tool tip
- export picture
- ruler tool

© 2026 Geomage 2003 LTD

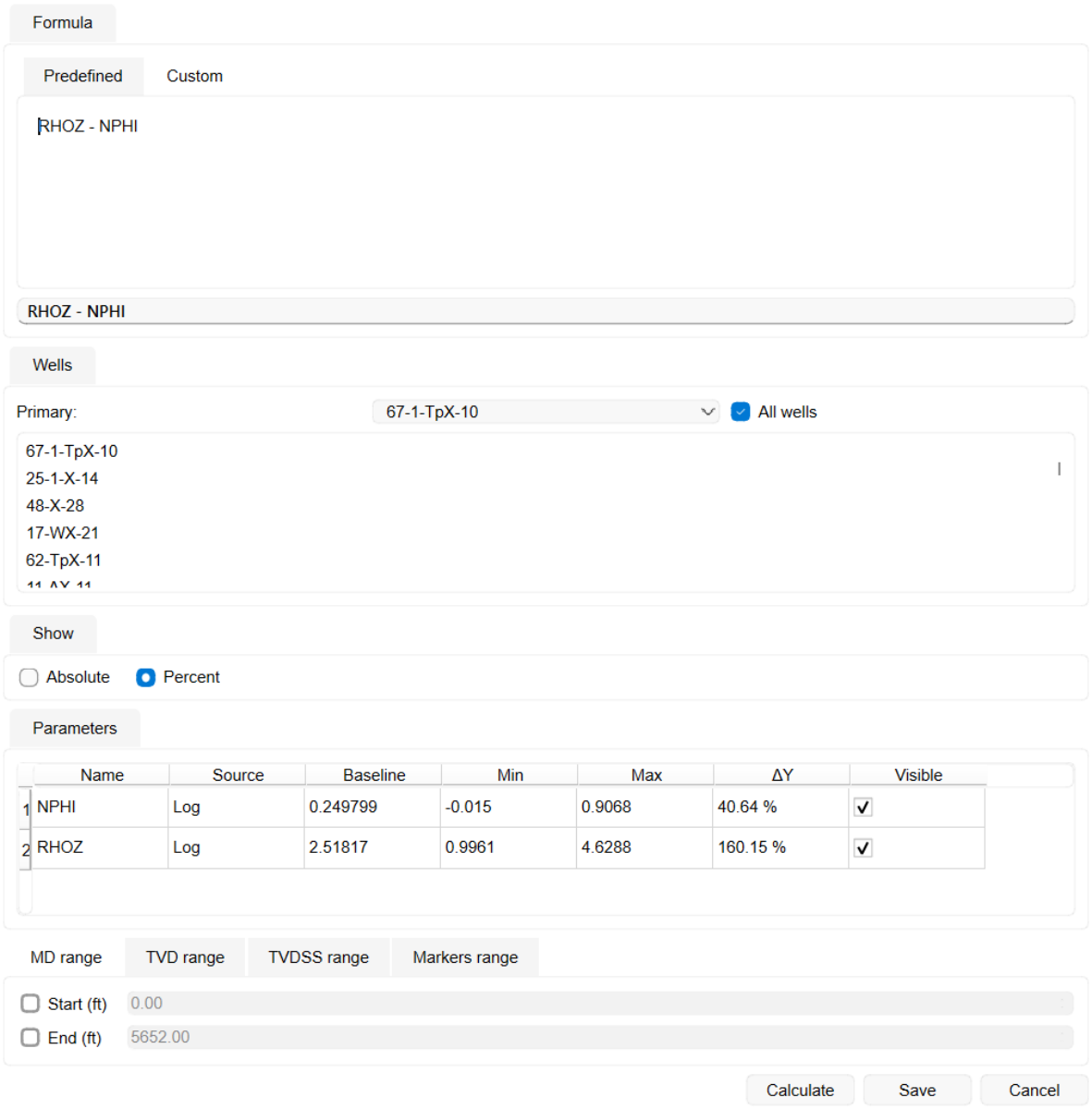
3.4.8.6 Tornado view

The **Tornado** view is a deterministic sensitivity analysis tool. A formula is first calculated with all of its input parameters at their baseline values, and then one parameter at a time is varied between its minimum and its maximum while the remaining parameters stay at their baseline. The length of each bar shows how much that single parameter changes the result, so the most influential parameters can be recognized at a glance.

To create a Tornado view go to the Views bar 38, navigate to **Graphs** and click on **Tornado**. The new view appears in the View manager 189 of the active workspace 224 and stays empty until a formula has been calculated.

To define the formula and its parameters, click **Formula** in the view functional panel. The **Tornado Formula** dialog opens:

© 2026 Geomage 2003 LTD



The dialog is organized in the following groups:

- **Formula** - select a ready-made expression on the **Predefined** tab (Porosity, Water saturation, Net-to-gross, Pore Radius or Permeability), or type your own expression on the **Custom** tab, using the same syntax as the Well log calculator 558. The expression that will be calculated is shown in the read-only field below the tabs.
- **Wells** - the well chosen in the **Primary** list provides the depth limits and the markers used by the range tabs. Select **All wells** to include every well of the project, or clear the check box and select one or several wells in the list below.
- **Show** - choose whether the sensitivity is expressed in **Absolute** output values or in **Percent** of the baseline result.
- **Parameters** - one row is created for every parameter name that is used in the formula.

© 2026 Geomage 2003 LTD

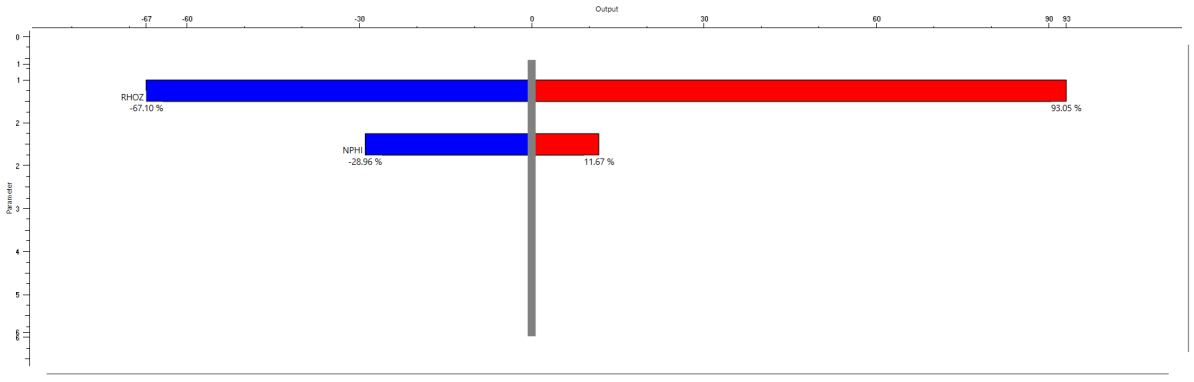
- The **MD range**, **TVD range**, **TVDSS range** and **Markers range** tabs at the bottom of the dialog optionally restrict the well log interval that is used for the calculation.

The **Parameters** table has the following columns:

- **Name** - the parameter name as it is written in the formula.
- **Source** - the data the values were taken from: **Grid**, **Log**, **Map** or **Constant**. When the same name exists in several data types, grid properties are used first, then well logs, then maps, and a message informs you when a grid property was used although a log curve of the same name exists. A name that is not found in the project is treated as a constant, and its values have to be entered manually.
- **Baseline**, **Min** and **Max** - the mean, the minimum and the maximum value found in the selected wells. Any of these cells can be edited, and a value that you type is kept for the following calculations.
- **ΔY** - the change of the result that is produced by this parameter, given in percent of the baseline result when **Percent** is selected.
- **Visible** - clear the check box to hide the parameter in the chart. The parameter stays in the table and in the calculation.

Click **Calculate** to fill in the table and refresh the chart while the dialog stays open. **Save** applies the calculated result to the view and closes the dialog, and **Cancel** closes the dialog and restores the last saved result. If the formula cannot be evaluated, or a parameter has no baseline or equal **Min** and **Max** values, a message describes the problem and the result is not applied.

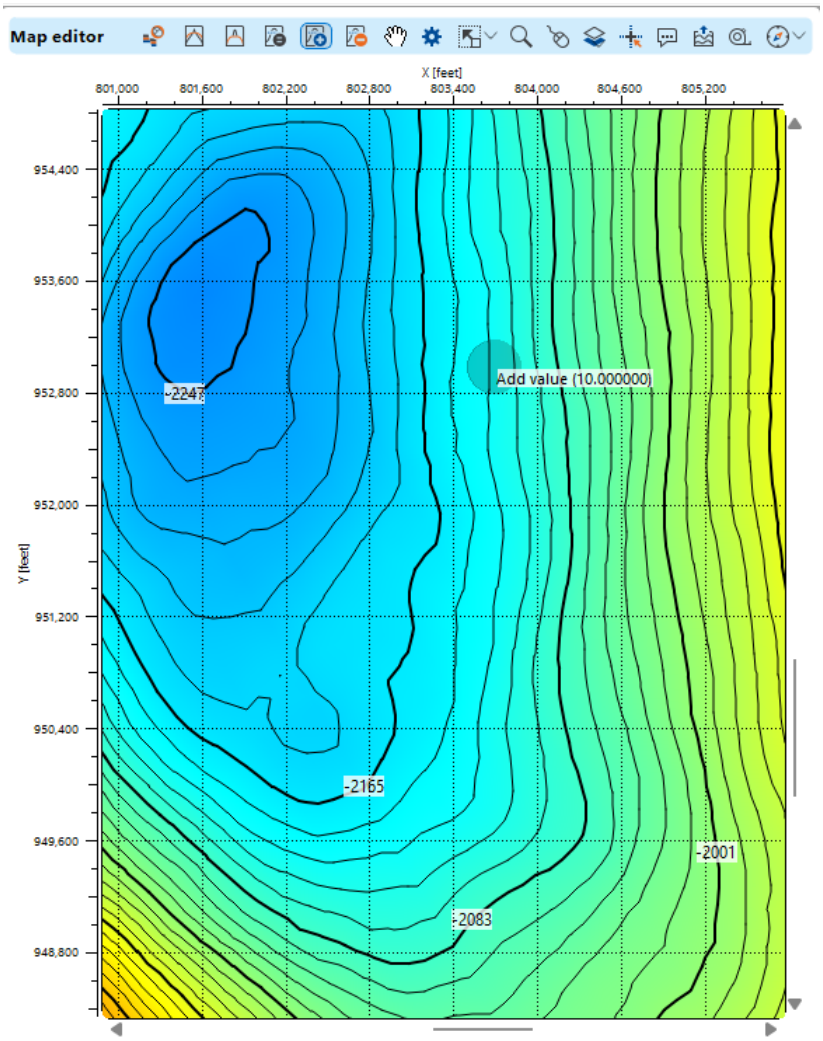
In the chart each row is one visible parameter, sorted from the strongest to the weakest influence. The vertical grey line marks the baseline result. The blue bar covers the range in which the result falls below the baseline and the red bar the range in which it rises above it, and the labels next to the ends of the bars give the corresponding values.



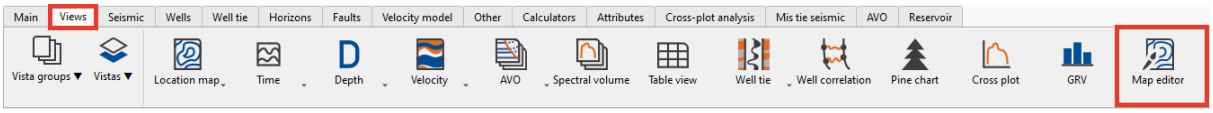
© 2026 Geomage 2003 LTD

3.4.9 Map editor

The **Map Editor View** in g-Space is a powerful tool for creating, editing, and analyzing contour maps and other spatial data visualizations. This view enables users to interactively modify and refine maps that represent various geological or geophysical properties.



To open **Map editor** view go to the Views bar 38 in the Ribbon bar 33 and click on editor view button:

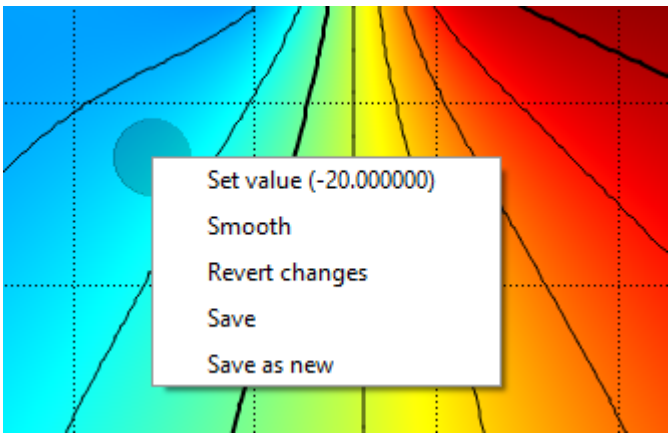


© 2026 Geomage 2003 LTD

The newly created view will appear in the View manager 189 in the active Workspase 224. User can place map editor view by dragging it to any workspace except pre-defined once (Seismic data 228 and Well data 232).

To define the object visibility stile go to the Visual Settings 206 by pressing in the view functional panel or find it in the panel 123 sets.

Important settings are located in the context menu (alt + RMB):



Here the user can:

- set value for the selected function
- smooth entire map
- revert all changes applied
- save edited map instead of original one
- save edited map as new one

Functional panel allows user to perform the following actions:

smooth

remove peak

© 2026 Geomage 2003 LTD

set value

add value

subtract value

open Visual Settings panel

adjust view area to all visible objects

adjust horizontal scale

adjust vertical scale

set/unset zoomable

set control item

add Object

enable mouse tracker

enable tool tip

export picture

ruler tool

show compass

show minimap

show scale bar

3.5 Controls and shortcuts

Mouse and keyboard controls

Buttons	Result	Applicable for:

© 2026 Geomage 2003 LTD

Left click mouse	pick the active object	Geo-view, Well-view
Right click mouse on view	delete, remove the picked object	Geo-view, Well-view
Right click mouse on panel	launch additional functionality	Data manager, Workspace manager, Module properties
Double click	making the object active	On data manager
Scroll button	zoom in/zoom out	Geo-view
Scroll button on axis	zoom in/zoom out the selected axis	Geo-view, Well-view
Pressing scroll button and moving mouse	dragging the view	Geo-view, Well-view
Alt + Right click mouse	launch additional functionality according to the active picking mode	Any of seismic view
Ctrl + Left click mouse	select multiple objects	Data manager

In g-Space you can use shortcuts to make your routine work faster and convenient

Action	Shortcut	Description
Update horizon	m	Lunching the map interpolation algorithm on location map 235 view or both Time and Depth seismic views 240
flattening on marker	f	Align the well section along the marker on 'well correlation view'
Save project	Ctrl+S	Saving the project 359, creating new project 357
Save project as	Shift+Ctrl+S	Saving the project 359 to a different file
Full screen	F11	Switch the use of all monitors connected to workstation. Addition click will put application to prevision screen setup.
Reset GUI settings	Ctrl+Alt+R	Reset settings of the interface to g-Space default state.
Show capture view	Ctrl+Alt+P	
Database connection	Ctrl+Alt+D	Open the database connection dialog
Help	F1	

3.6 Seismic data

Seismic data is the main display window where the user can able to visualize the entire dataset within a window. The entire dataset is loaded into the memory and display the full content of

© 2026 Geomage 2003 LTD

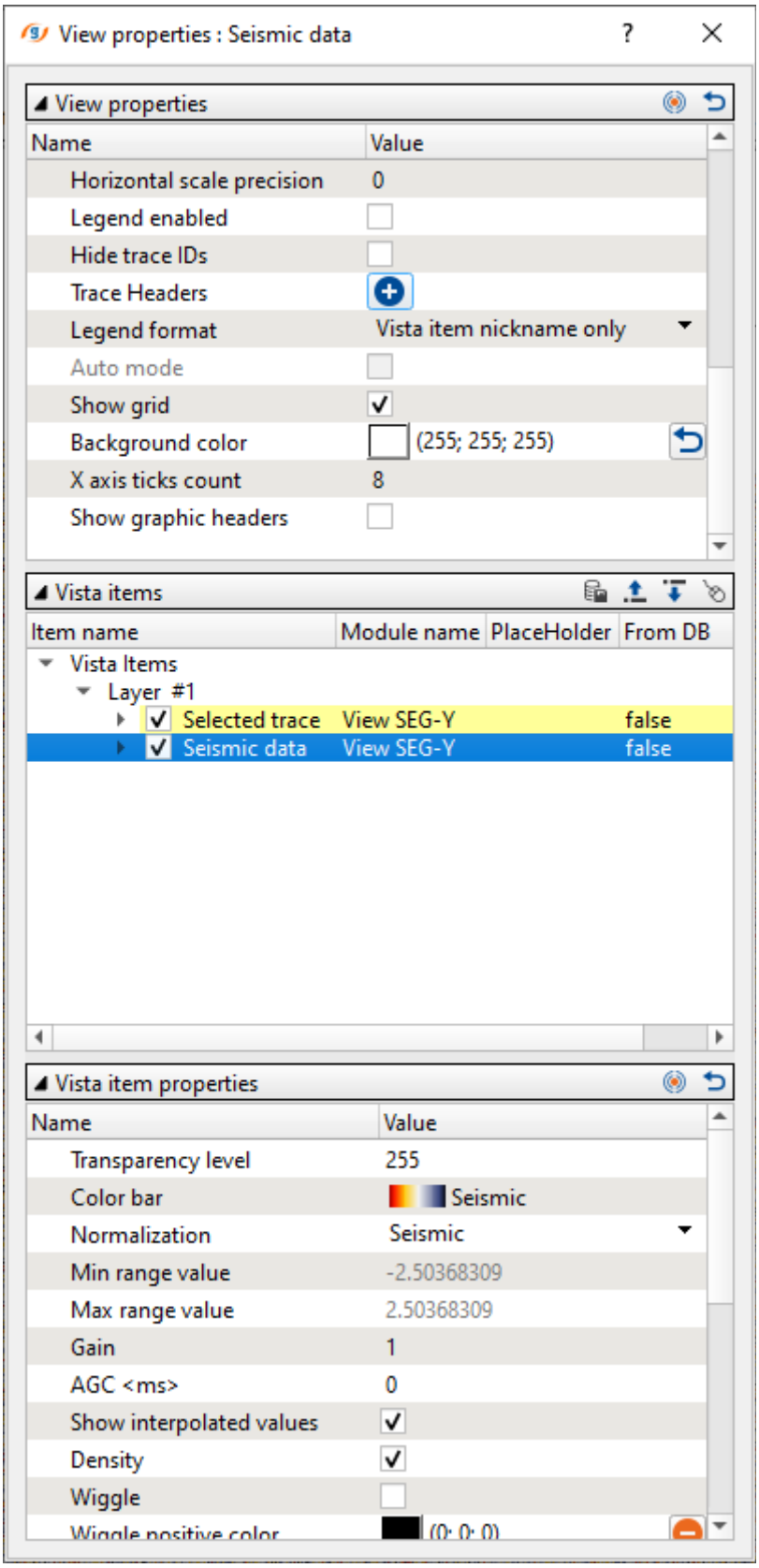
the data in the seismic data window. However if the user zoom in/out in any part of the **Seismic data** then that particular section of the display will be displayed on the screen.

Within the seismic data window, we have many option to visualize the data. For an example if the user wants to display the Offset plot on the top of the annotations, it can be done fairly easy. Before that we need to introduce various icons on the top of the seismic data window each got it's own significance.

1. Show properties

This will open up a pop-up window to change the display settings like placing annotations on the top, changing the color of the display from density to wiggle, applying AGC etc.

© 2026 Geomage 2003 LTD



© 2026 Geomage 2003 LTD

Show properties is designed into 3 parts.

Visual Settings predominantly takes care of Trace headers annotations , showing graphic headers etc.
Object items deals with the main components of the data like seismic data, selected trace, selected shot gather etc
Object item properties helps in adjusting the gain, AGC, change the color bar, applying Band pass filter etc.

2. Adjust to size **:**

This option adjust the display size into the original position. Zoom Out.

3. Set/Unset zoomable :

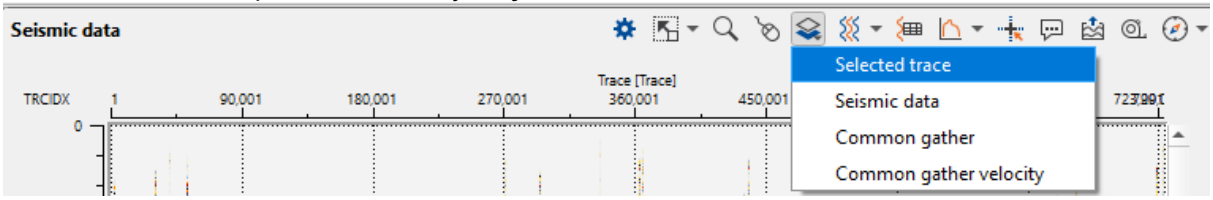
If the user wants to zoom a particular area, simply click on the icon to activate the zoom and draw a polygon on the seismic data to zoom that particular area. Please note that after the initial operation, the icon resets to zoom out position and the user needs to reselect the **set/unset zoomable** icon to zoom again.

4. Set control item :

This icon helps in selecting any particular item inside the seismic data window. For instance, if the user wants to select a trace then click on **Set control item** icon and it will give the options to choose. In this instance we have only one option i.e. select trace.

5. Add Object :

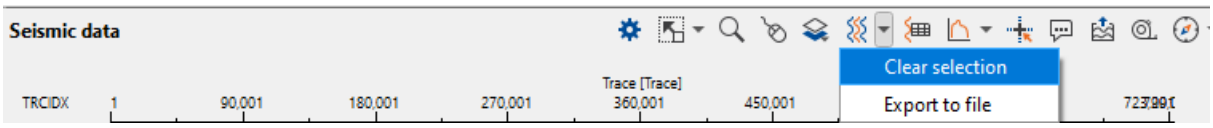
This icon helps us to add any Object items like as shown below.



6. Select traces **:**

© 2026 Geomage 2003 LTD

By selecting this icon, the user can able to select a trace. Besides this, we have an option to clear to Clear the selection and Export to file option if the user click on the downward triangle right next to the Select traces icon.

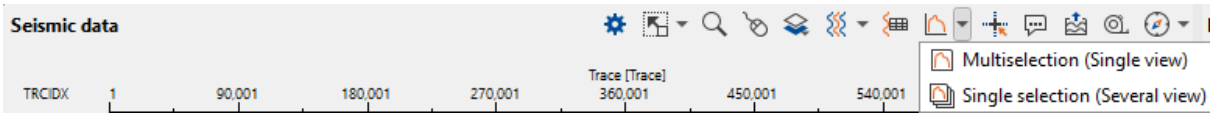


7. Trace headers table **:**

This option will work in tandem with the Select traces module. The trace headers information should be available in the Trace data window.

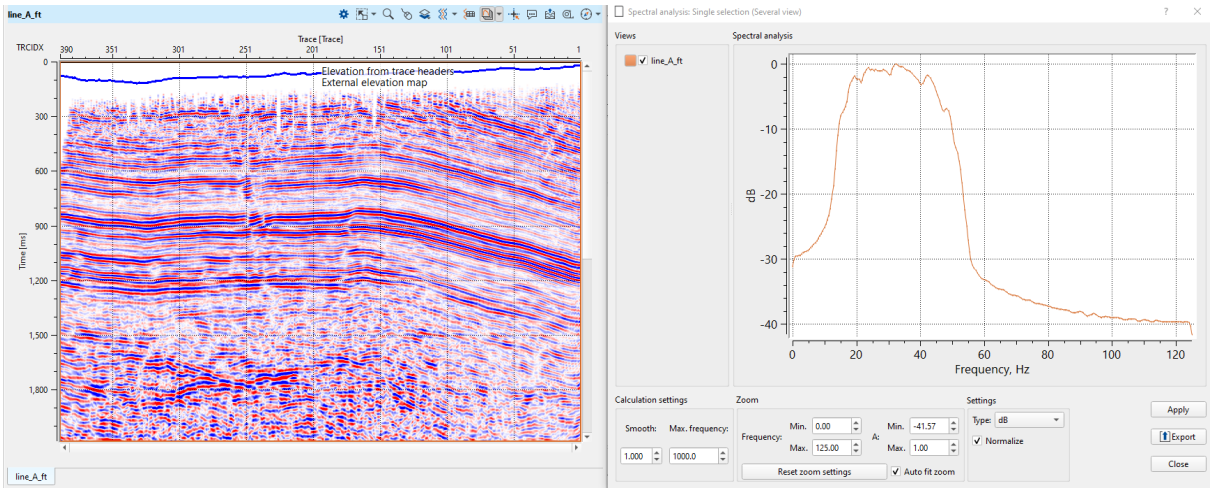
8. Spectral Analysis **:**

Within g-Viewer the user has the ability to analyze the amplitude and frequency of the seismic data via a spectral analysis. To perform this the user should go to either the Seismic data or the Common shot/receiver/bin tab.
Within the Spectral analysis icon, we have two options. **Multiple selection** (single view) and **Single selection** (Several view). By default, it will display the Single selection (several view).

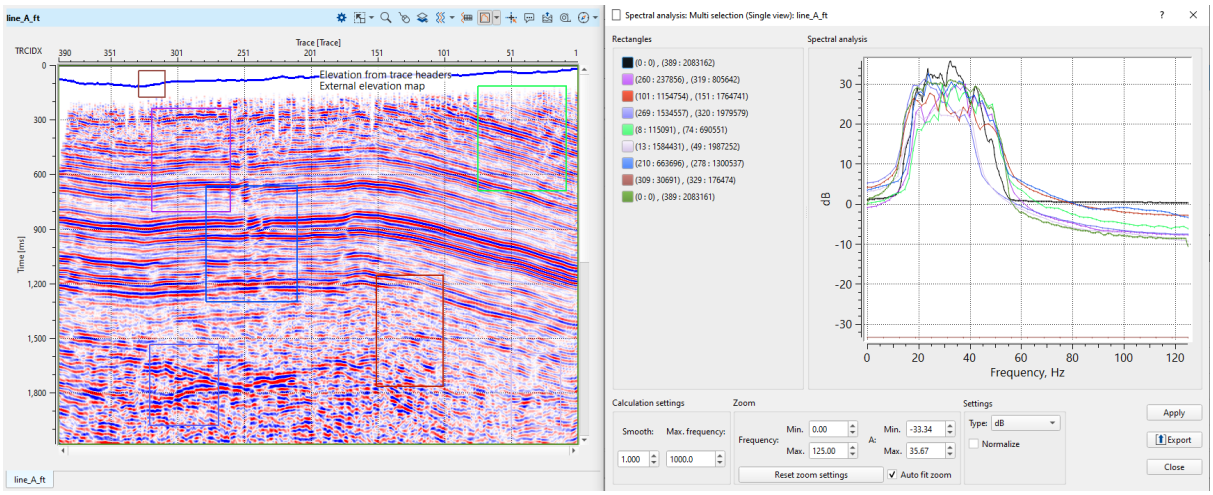


Click on the **Spectral analysis** icon and it will generate the spectral analysis window as shown below. By default Single selection is selected and the corresponding spectrum is displayed. In the below image, we can see that the entire stack section is selected and the corresponding spectrum is displayed in a separate window.

© 2026 Geomage 2003 LTD



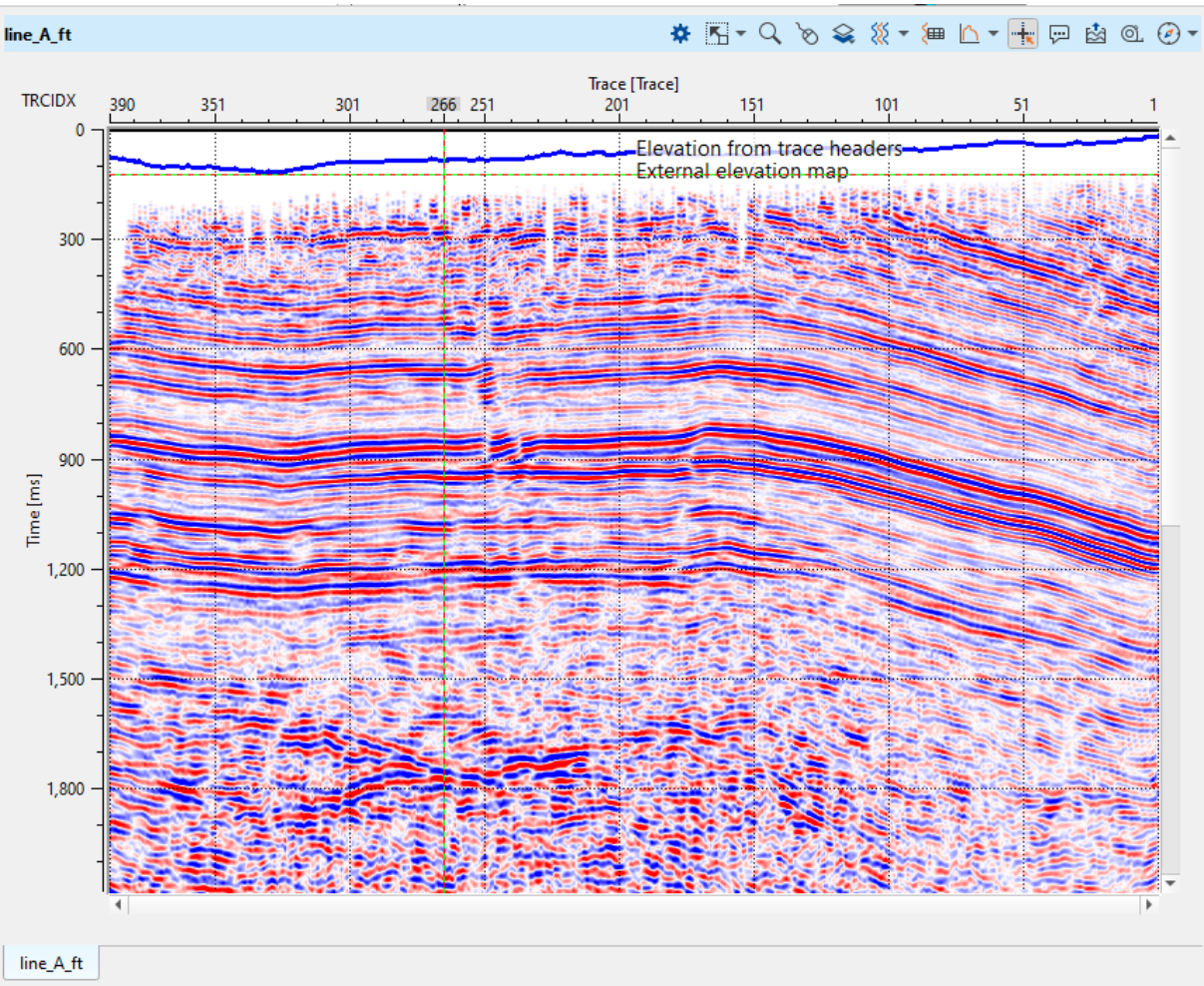
If the user wants to analyze different areas in a display then select the "**Multiple selection**" option and draw as many areas as desired to display the spectral analysis. To draw a polygon, simply hold and drag MB1 button at the desired location on the display window. Several window selections and their corresponding spectra are displayed below.



9. Set/unset mouse tracker **:**

When the user selects this option, it will show the dotted horizontal and vertical lines to show the mouse tracker as shown below.

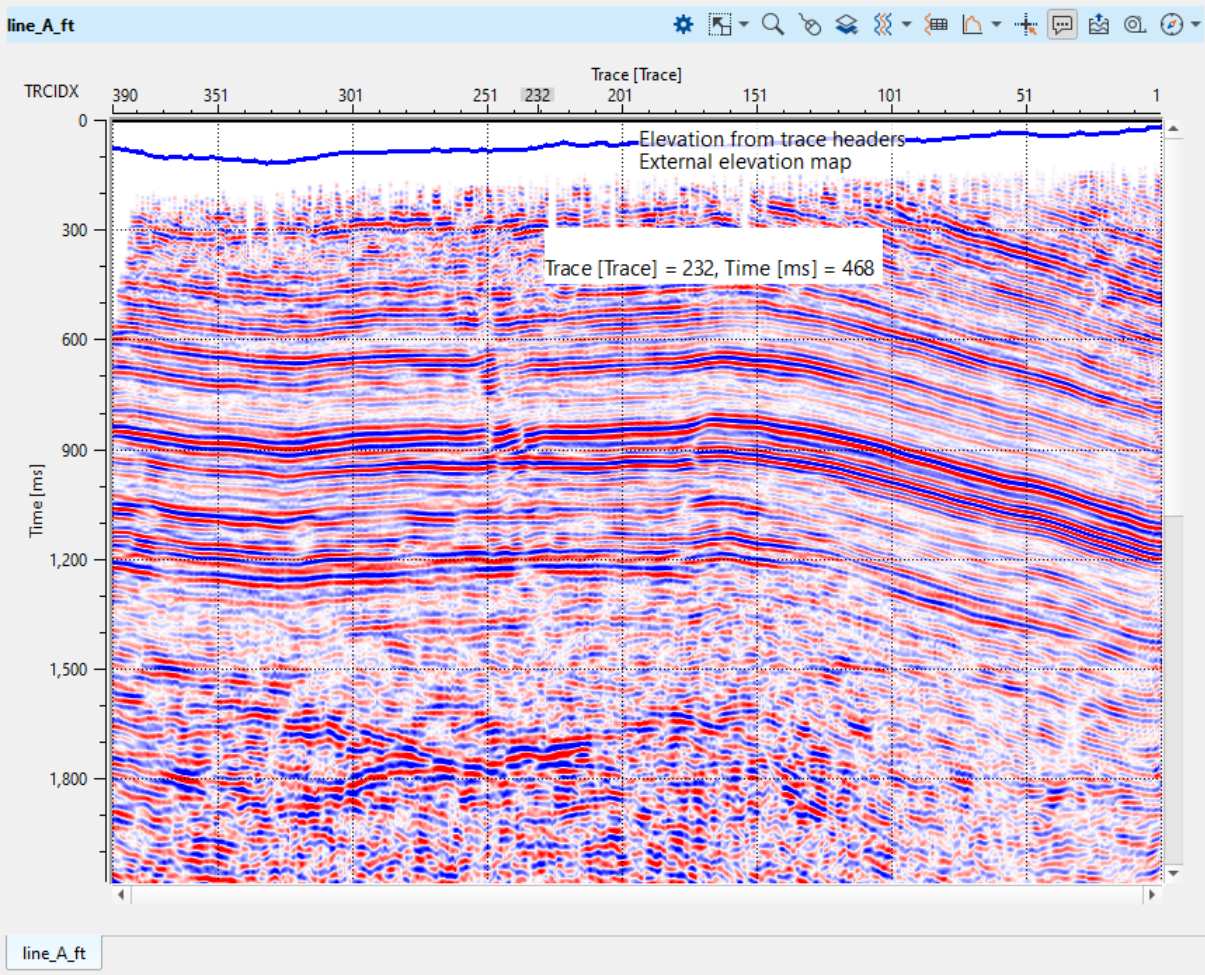
© 2026 Geomage 2003 LTD



10. Tool tip **:**

Upon selecting this option, it will display the available information depending on the cursor position.

© 2026 Geomage 2003 LTD



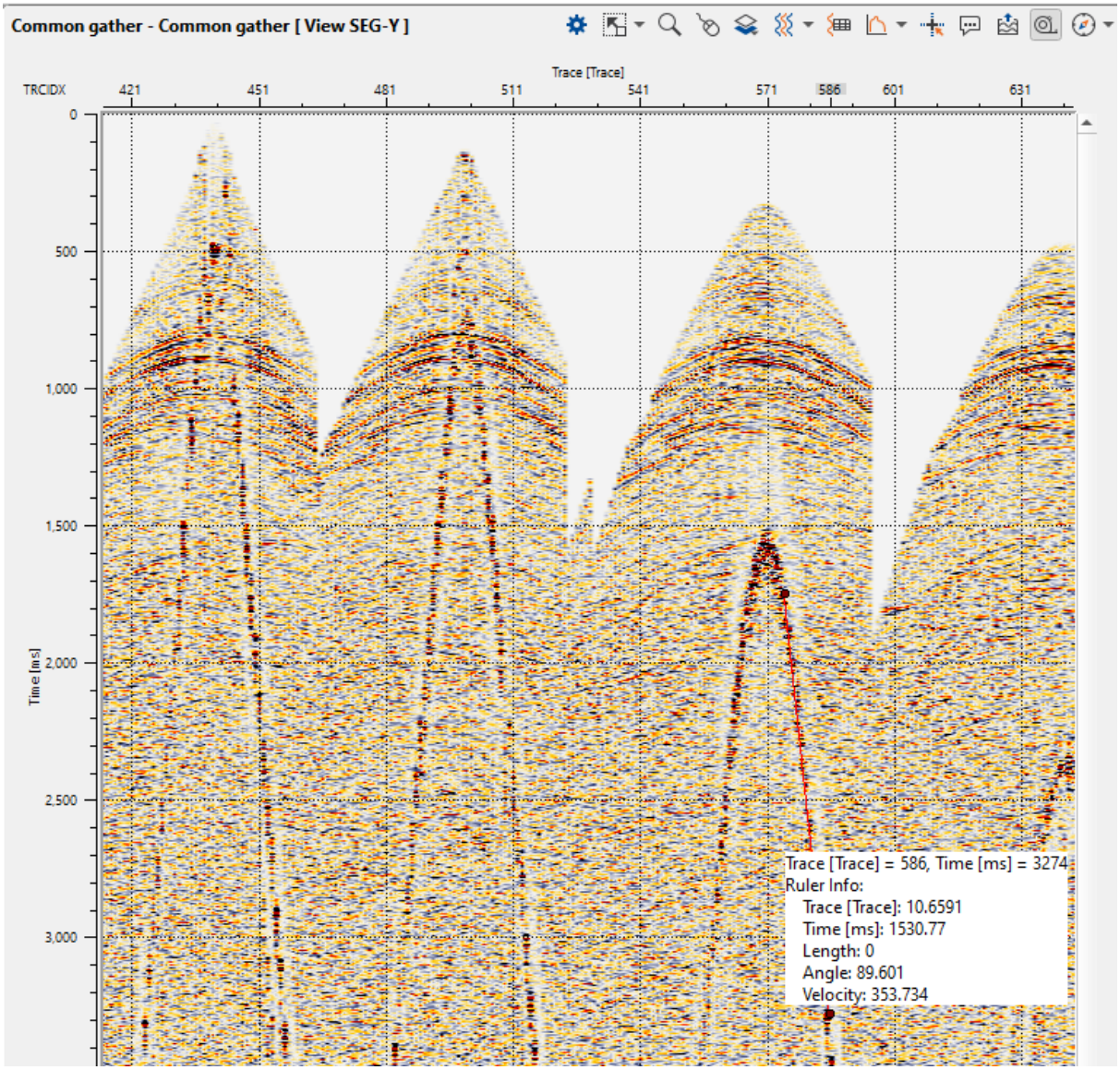
11. Export picture **:**

The user can able to import the images with different formats for future needs like power point presentations etc.

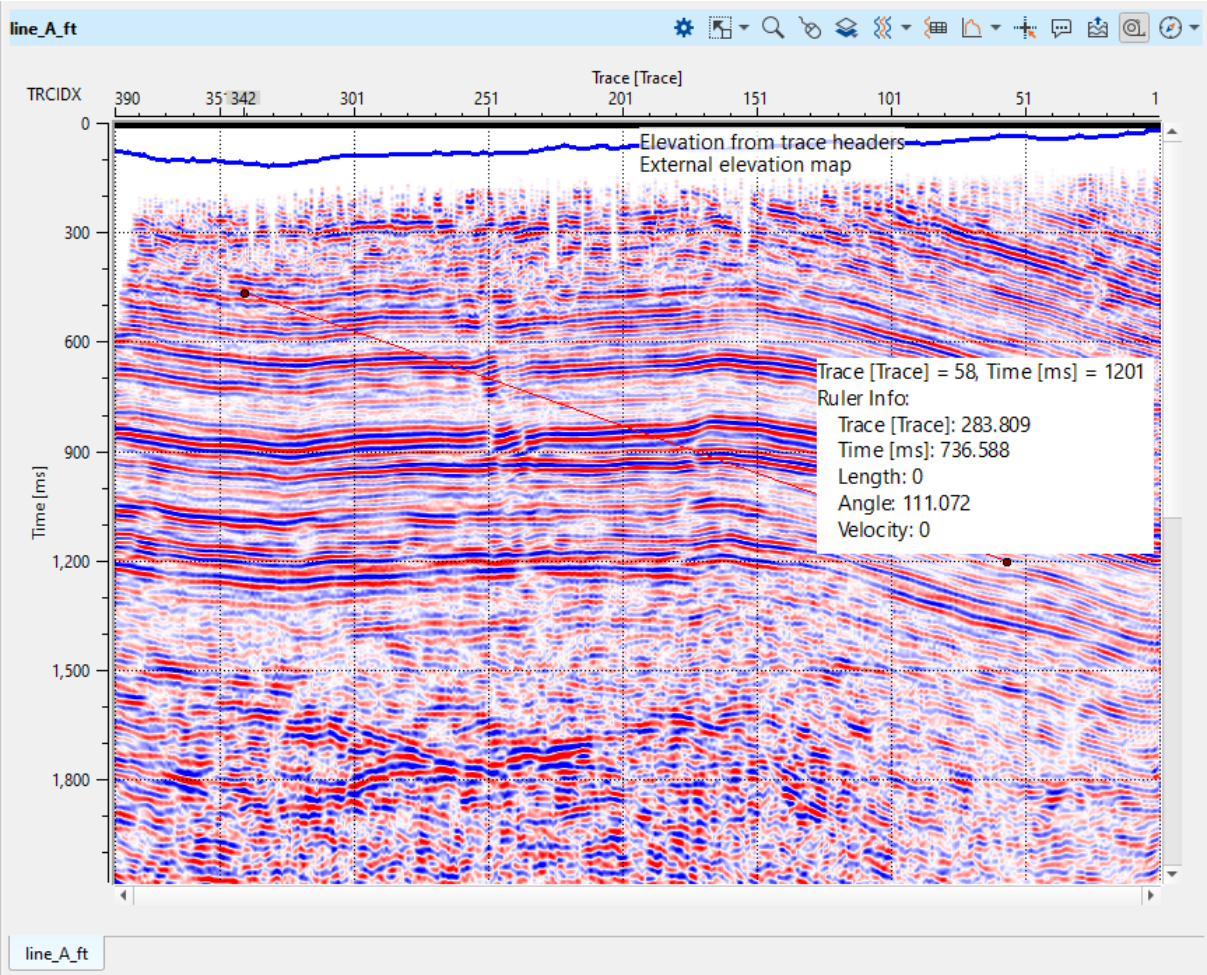
12. Ruler **:**

This is a very useful tool especially for **Prestack** data. For an example, we want to measure the velocity of an event. To achieve click on the Ruler icon and hold MB1(left mouse button) and draw a line. It will display the information as a tool tip as shown below. In case of post stack data, it isn't that informative.

© 2026 Geomage 2003 LTD

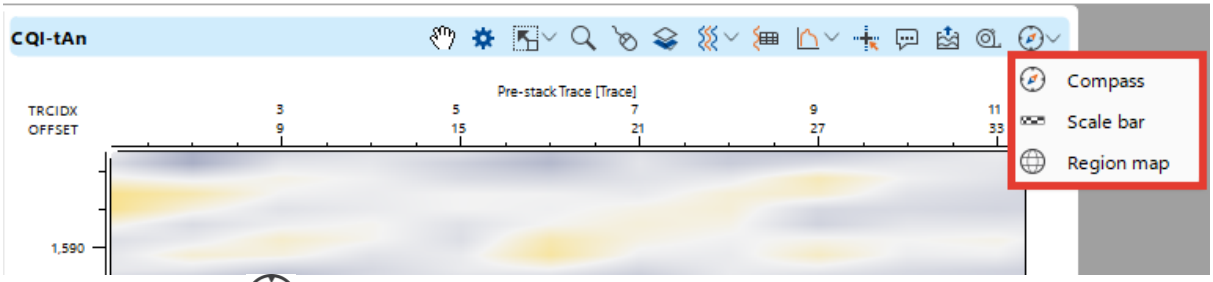


© 2026 Geomage 2003 LTD



13. Miscellaneous

The *Miscellaneous* icon provides additional tools that enhance the seismic data interpretation experience. These tools add orientation, scale, and context to the data being visualized, especially in spatially complex environments. The main components include:



- **Compass** Displays an orientation guide within the seismic data window, helping users to maintain a clear sense of direction.

© 2026 Geomage 2003 LTD

- Scale bar Adds a scale reference to the display, allowing the user to visually gauge distances across the seismic section.

- Region map Shows a miniaturized map view of the region or data coverage area, providing spatial context for the seismic data being analyzed.

© 2026 Geomage 2003 LTD

Getting Started

4 Getting Started

This chapter will guide you through the essential steps of using g-Space software.

You'll find detailed instructions on installation 14 and licensing 23, as well as how to create 357, save 359, and load 361 projects. Whether you're starting a new project or working with an existing one, this section covers all the basics.

In addition, you'll learn how to configure your project settings 363, including selecting the appropriate measurement systems supported by g-Space.

We also provide an overview of the demo project 369 included with the software, which offers a hands-on opportunity to explore g-Space's capabilities with preloaded data.

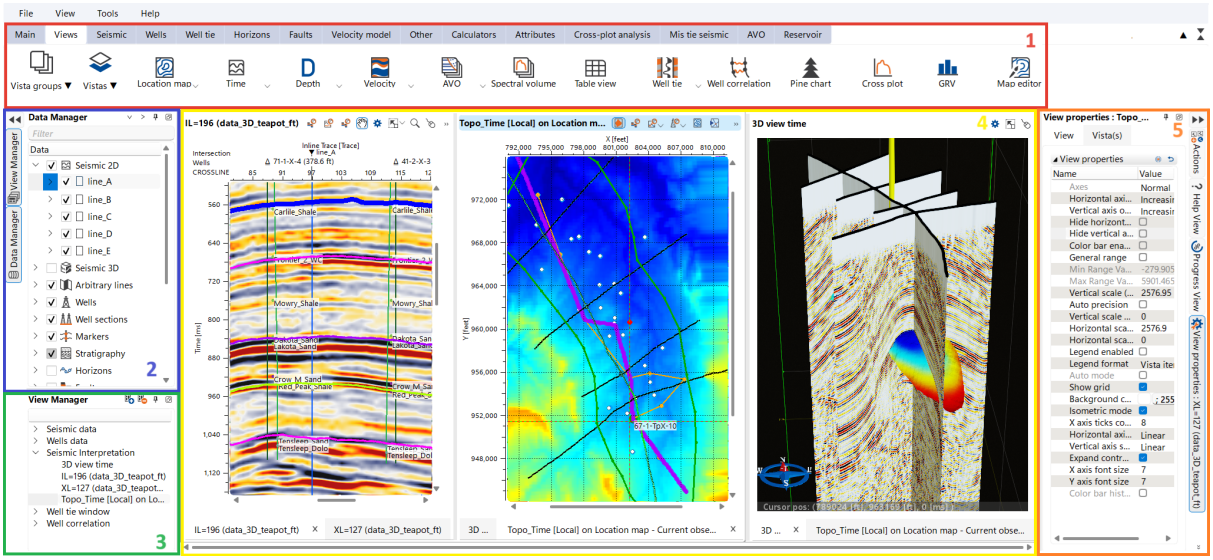
g-Space is a powerful platform designed specifically for seismic interpretation and geomodeling, offering geoscientists advanced tools to analyze and interpret subsurface data with precision.

As the second software package in our product line, g-Space complements g-Platform™ — our leading solution for seismic processing and imaging. While g-Platform™ handles the initial stages of data acquisition and seismic processing, g-Space takes over to drive the workflow from seismic interpretation to comprehensive geological modeling.

In g-Space, seismic interpretation is at the core, with state-of-the-art algorithms and visualization techniques that allow users to accurately identify structural and stratigraphic features, evaluate reservoir quality, and understand properties distribution. The platform excels in integrating seismic data with well logs, stratigraphy, and other geological information, enabling the creation of detailed 3D geological models.

By the end of this chapter, you'll be equipped with the foundational knowledge to navigate g-Space, configure your projects, and begin leveraging its powerful tools for seismic interpretation and geomodeling, building upon the groundwork laid by g-Platform™.

© 2026 Geomage 2003 LTD



4.1 Creating a new project

To **create new project** follow next steps:

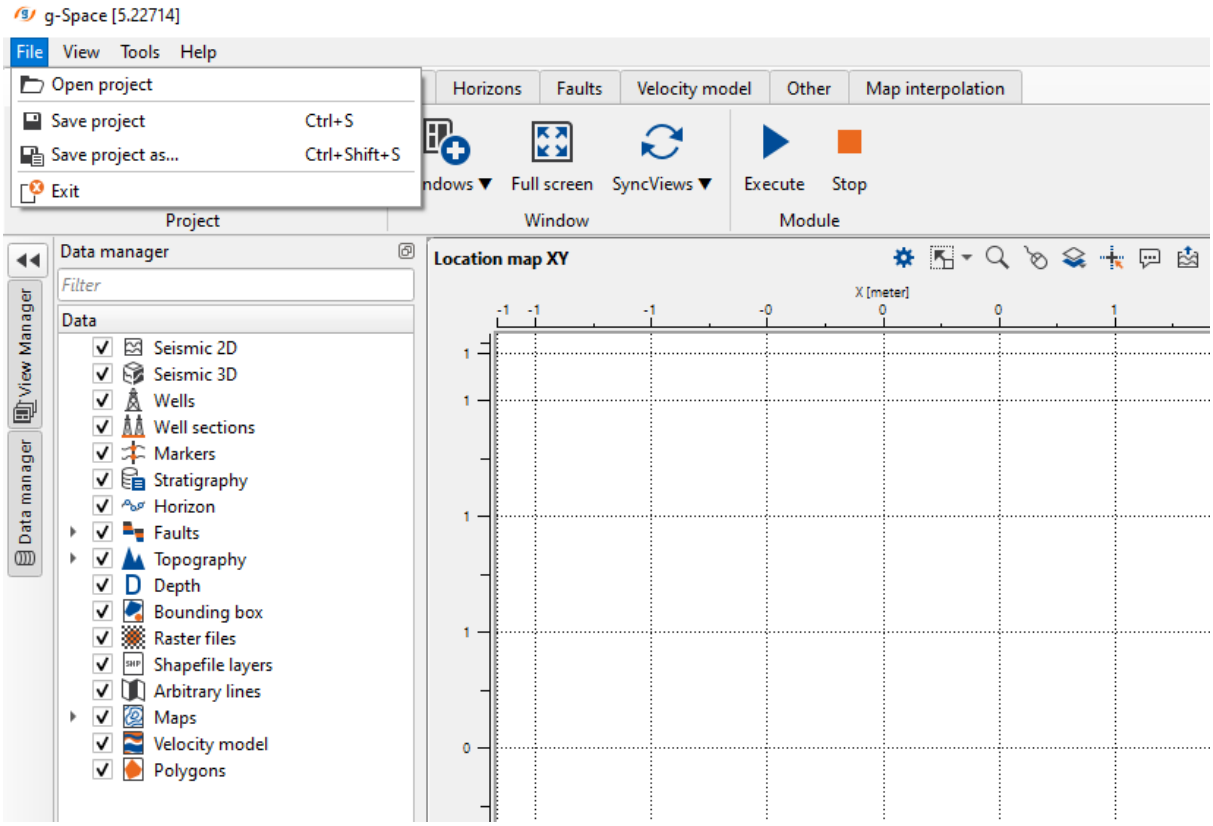
When g-Space starts it already contains an empty project. A new empty project can also be started at any time from the Menu bar with **File > Create new project**. All data containers (the categories shown in the Data Manager 124) are created automatically – the new project then only needs to be saved and configured.

Step 1 Open g-Space

Step 2 In Menu bar click - **File** and then **Save project** or

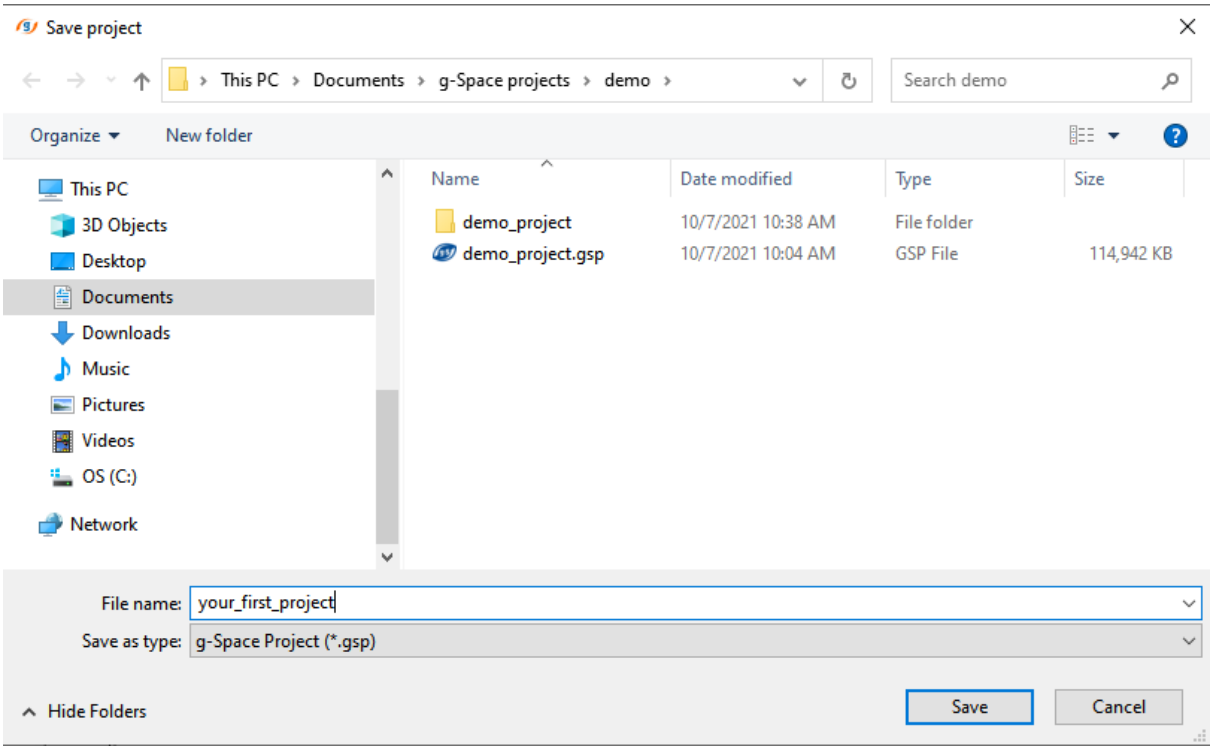
in Ribbon bar 33 press **Save project** or just press **Ctrl+S** shortcut 343

© 2026 Geomage 2003 LTD



Step 3 In the appeared window **Save project** set the destination of the file directory and click **Save** button.

© 2026 Geomage 2003 LTD



Step 4 Define the project coordinate system and measurement units in the Configuration 363 dialog (Main bar > Configuration). See Coordinate and measurement system 365.

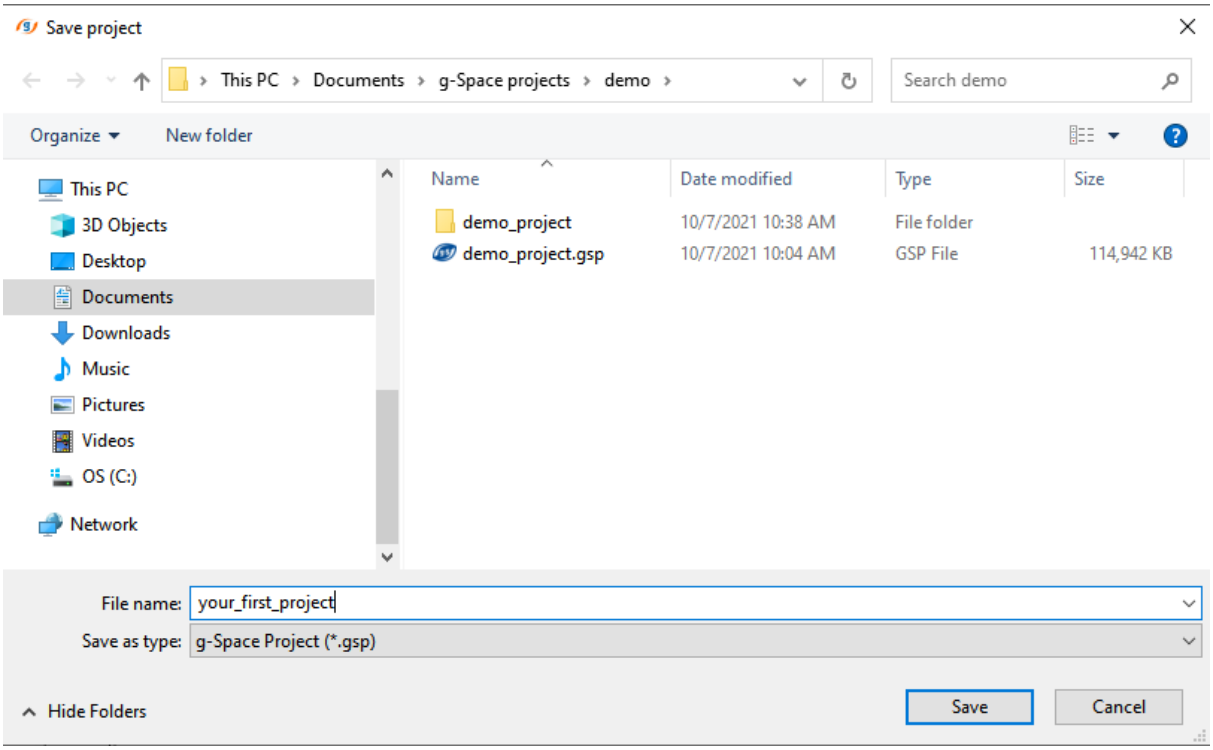
4.2 Saving the project

If you want to save changes to the existing project, use **option 1**.
If you prefer not to overwrite the current project and want to save it as a new file instead, use **option 2**.

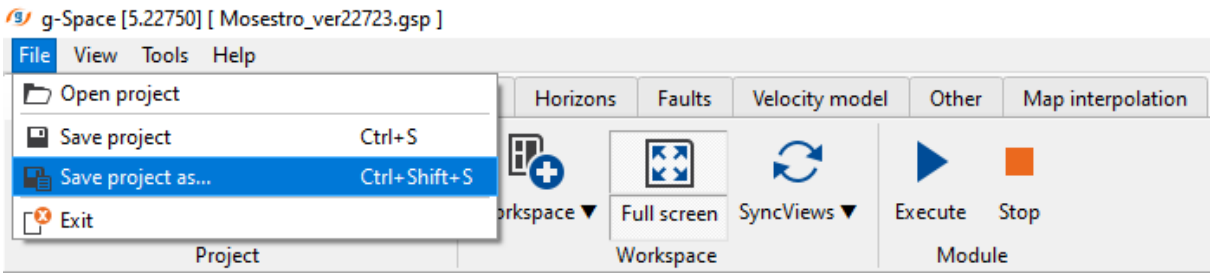
Option 1.
In Menu bar click **File** and then **Save project** or

in Ribbon bar 33 press **Save project** or just press **Ctrl+S** shortcut 343

© 2026 Geomage 2003 LTD

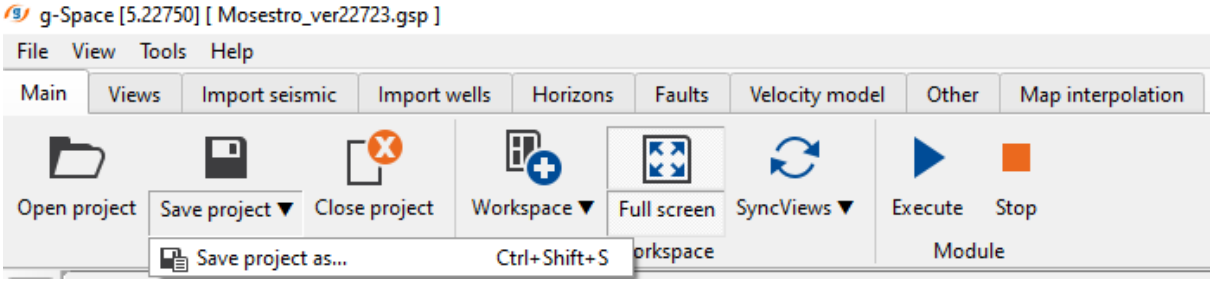


Option 2.
In Menu bar click **File** and then **Save project as...** or just press **Shift+Ctrl+S** shortcut 343



or in Ribbon bar 33 press on the arrow on the right bottom corner of the button **Save project** or just press **Shift+Ctrl+S** shortcut 343

© 2026 Geomage 2003 LTD



Depending on project size it may take from few seconds to few minutes to save the project.

Save project stores all the project configuration, data and views. Seismic data is referenced by file path and is not copied into the project. **Save project as** creates a copy of the project at a new location.

4.3 Loading an existing project

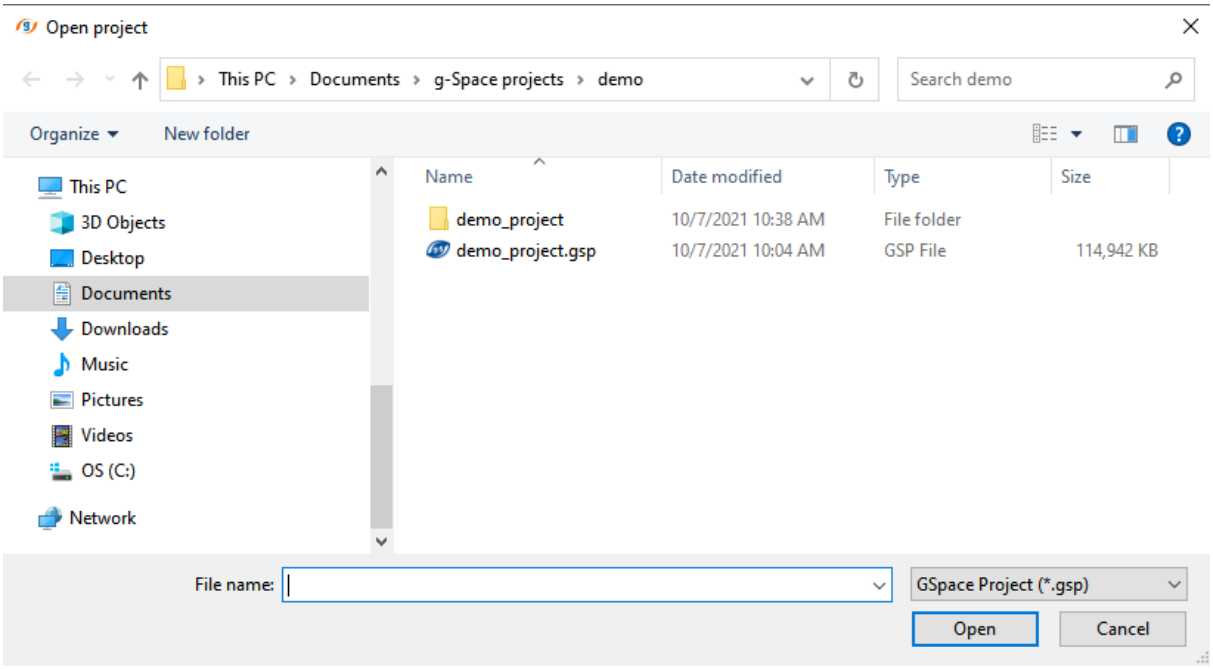
g-Space project consists of two files with the following extensions:

***.gsp** - stores the data loaded in the project
***.gsp.gui** - stores visualization settings you made in the project

To load the existing project in Menu bar click **File** and then **Open project** or in Ribbon bar 33 press **Open project**

In appeared window locate the file with extension ***.gsp** and click **Open**

© 2026 Geomage 2003 LTD

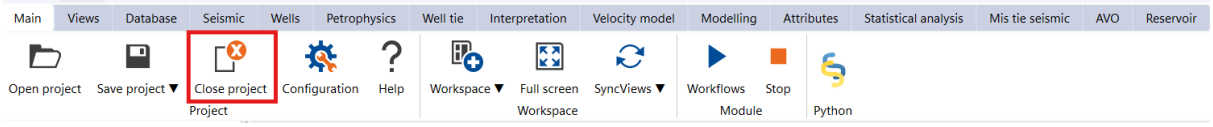


Depending on project size it may take from few seconds to few minutes to load the project.

4.4 Closing the project

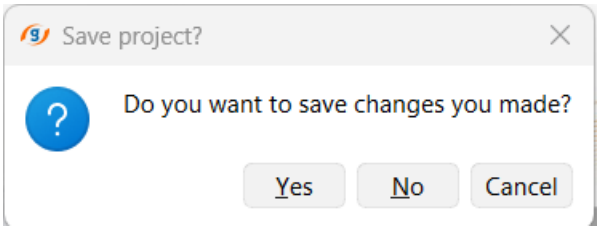
If you don't want to close g-Space, but want to close the current project, use the **Close project** command. Closing a project resets g-Space to a clean, empty project without exiting the application – it is the same operation as creating a new project 357.

In the Ribbon bar 33, on the **Main** tab, in the **Project** group, press **Close project**.



If the project contains unsaved changes, g-Space asks whether you want to save them. Choose **Yes** to save before closing, **No** to discard the changes, or **Cancel** to keep working in the current project. After closing, a clean unsaved g-Space project is opened.

© 2026 Geomage 2003 LTD



See Also

- Saving the project 359
- Creating a new project 357
- Loading an existing project 361

4.5 Project configuration

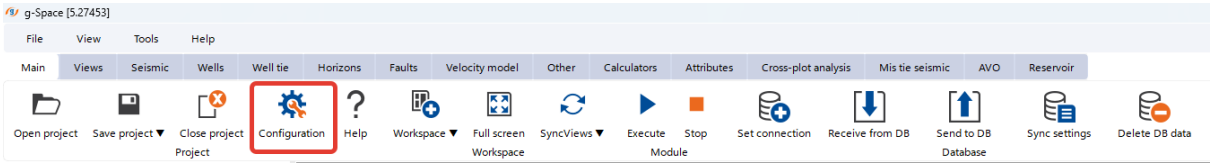
The **Project configuration** dialog stores the global settings of a g-Space project – the coordinate reference system, the measurement units, the datum mode, backup behaviour and the base directory. Configure these settings once, right after creating a new project 357, before importing data.

This chapter covers how to make global settings for the project such as:

- Setting up datum 368
- Setting up coordinate and measurement system 365

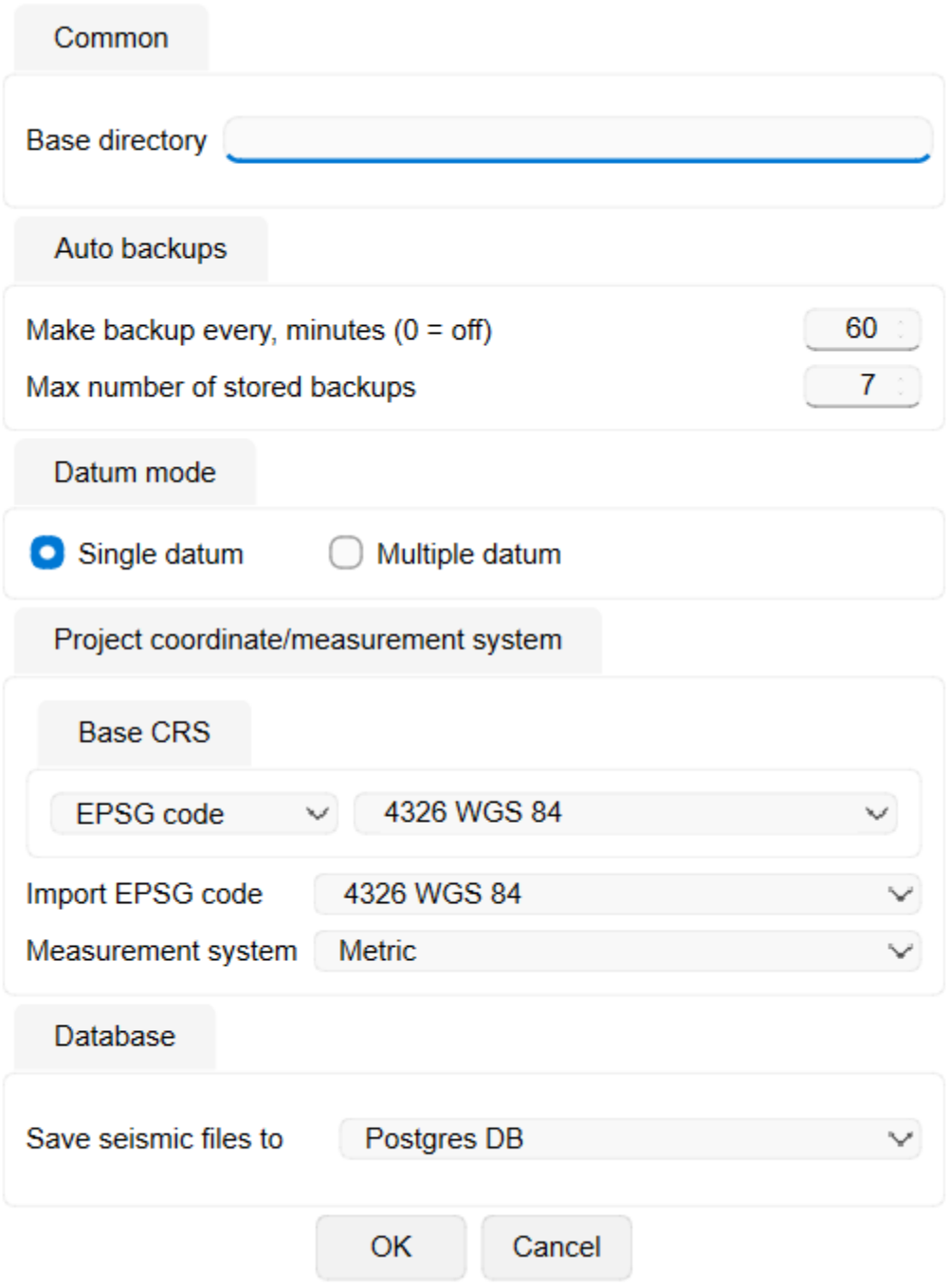
Opening the configuration dialog

Open the dialog from the **Main** tab of the Ribbon bar 33, in the **Project** group, by clicking the **Configuration** button. The same command is available from the Menu bar with **File > Configuration**.



The **g-Space project configuration** dialog opens:

© 2026 Geomage 2003 LTD



Configuration options

The dialog groups the project-wide settings into the following sections:

- **Common** – the project **Base directory**, the folder used to resolve relative data paths.

© 2026 Geomage 2003 LTD

- **Auto backups** – how often a backup is written (**Make backup every, minutes**; set to **0** to switch backups off) and the **Max number of stored backups**.
- **Datum mode** – choose **Single datum** (all seismic data use the same datum and replacement velocity – the most common case) or **Multiple datum** (seismic data have different datums or replacement velocities). See Datum configuration 368.
- **Project coordinate/measurement system** – set the **Base CRS** (searchable by EPSG code), an optional **Import EPSG code** for data in a different system, and the **Measurement system**. See Coordinate and measurement system 365.
- **Database** – the **Save seismic files to** mode, controlling where seismic files are stored when the project is saved to a database.

Click **OK** to apply the settings, or **Cancel** to discard them.

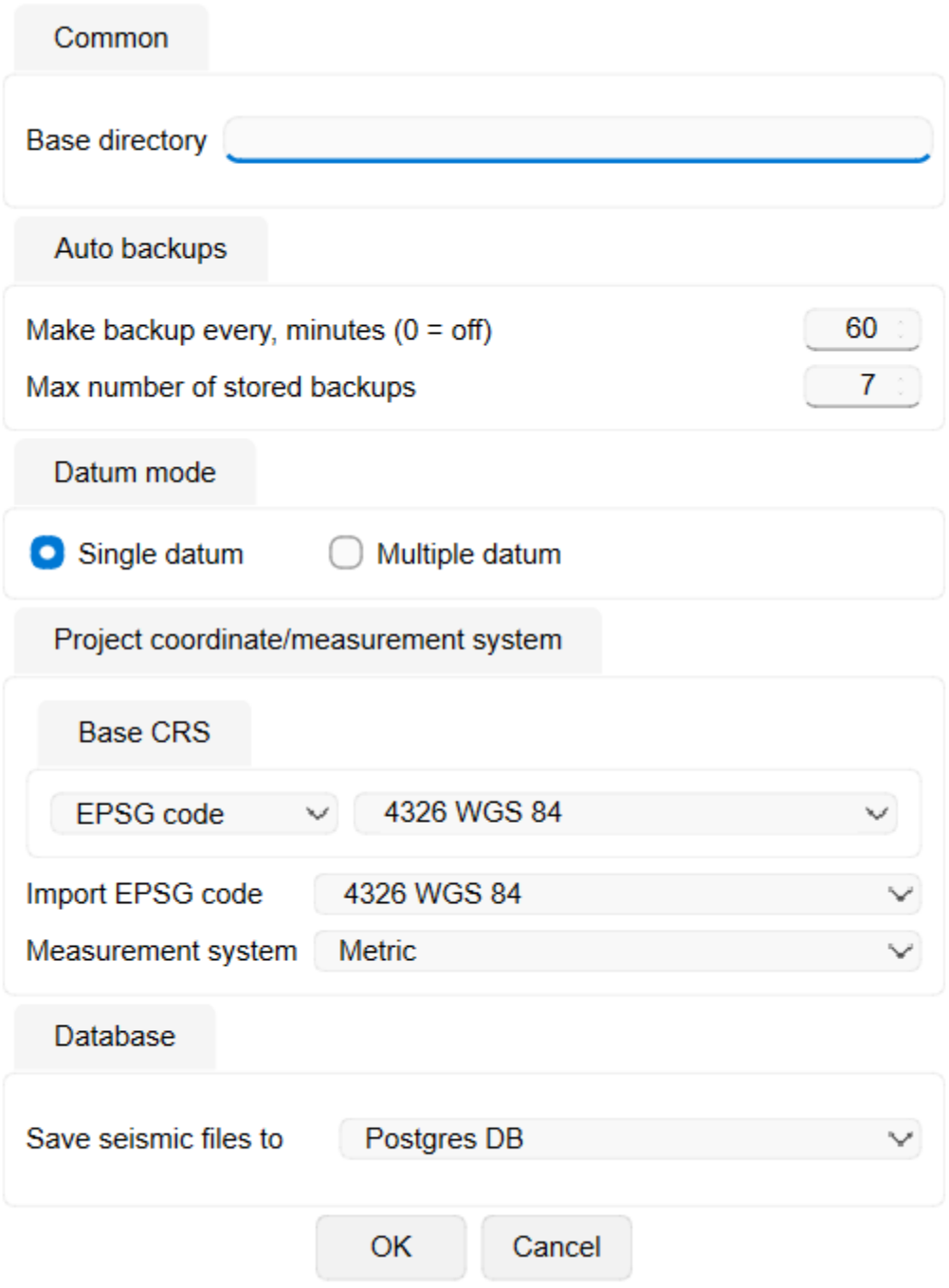
See Also

- Coordinate and measurement system 365
- Datum configuration 368
- Creating a new project 357

4.5.1 Coordinate and measurement system

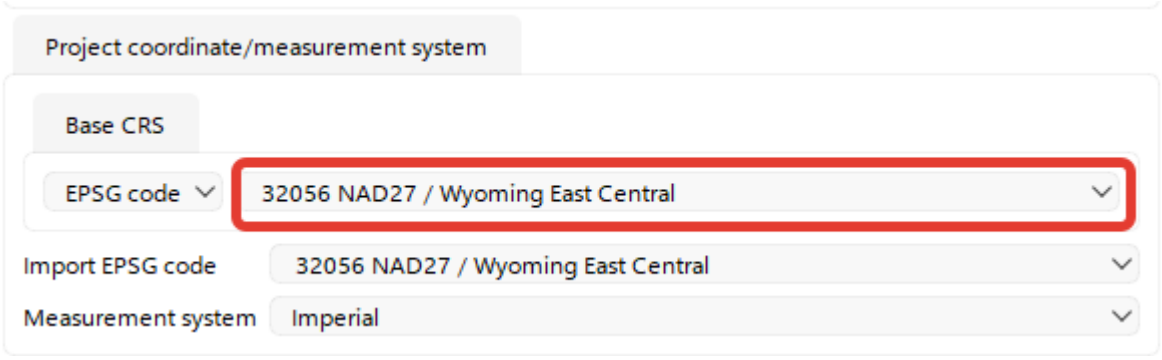
To configure the project's Coordinate Reference System (CRS), open the Project configuration 363 menu.
In this menu, you can set the CRS and choose the measurement system in the Project coordinate/measurement system section:

© 2026 Geomage 2003 LTD

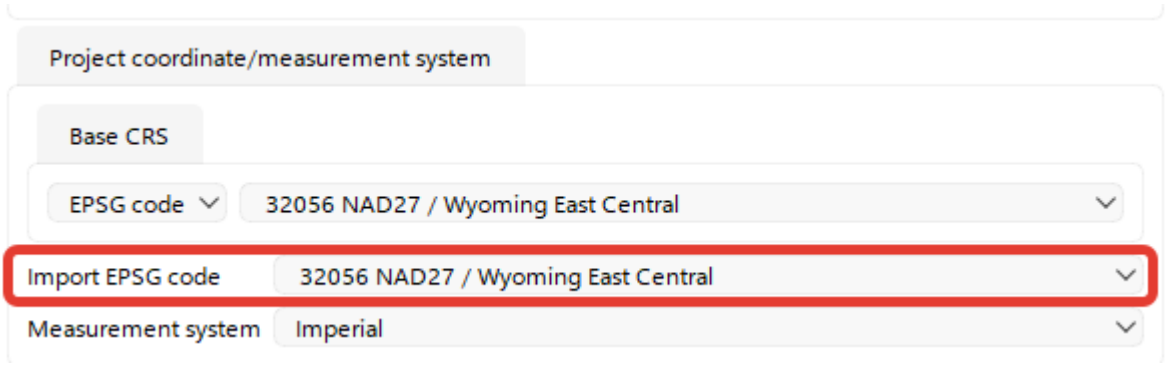


g-Space offers a vast selection of predefined CRS.
The easiest way to find the one that suits your project is by entering its EPSG code into the search field:

© 2026 Geomage 2003 LTD

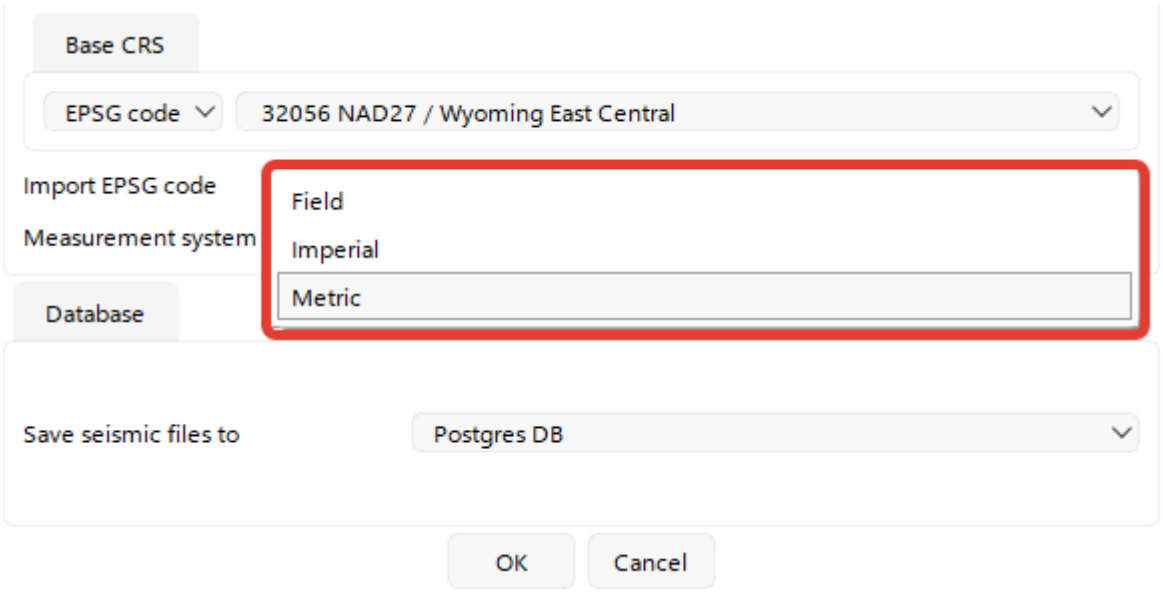


If the CRS of the imported data differs from the project's CRS, you can specify a different EPSG code for the import:



In this section, you can also choose the measurement system from three options: Imperial, Metric, or Field (coordinates in Metric and depth in Imperial):

© 2026 Geomage 2003 LTD



4.5.2 Datum configuration

The project datum defines the elevation reference and the replacement velocity used when reconciling data acquired or processed at different levels. Every depth- and time-domain calculation in g-Space — including horizon export, velocity modelling, and volumetric calculations — references this datum, so it must be correct before data is loaded or re-processed.

Setting the datum

To configure the project datum, follow these steps:

1. Open Module properties 197 and select the **Parameters** tab.
2. Set the required values for **Replacement Velocity** and **Datum**.
3. Tick the **Shift to Datum** checkbox so that subsequent imports and exports are referenced to the new datum.

Datum mode (single or multiple)

The project **Datum mode** controls whether all seismic data share one datum or may each use their own. Open it from the **File** menu > **Configuration**, in the **Datum mode** group:

- **Single datum** (default) — all seismic data use the same datum and replacement velocity (most common case). Seismic import then runs in a simplified mode with the per-file datum fields hidden.

© 2026 Geomage 2003 LTD

- **Multiple datum** — seismic data have different datums or replacement velocities; the seismic import view exposes per-file datum and V0 (replacement velocity) fields.

When to change the datum

- At project creation, to match the datum of the primary seismic survey.
- Before importing horizons or wells whose elevations are referenced to a different datum — use the Shift to Datum option to translate values on the fly.
- When merging data from surveys processed against different replacement velocities.

Changing the datum after data has been loaded causes every datum-dependent object to be recalculated — expect a service-task pass to follow (see Service Tasks 526).

See Also

- Module properties 197
- Coordinate and measurement system 365
- Creating a new project 357

4.6 Demo project

The demo project is included in the installation package and is located in the user folder: *C:\Users\"USERNAME"\.gPlatform_v4\GSpace\DEMO*

This demo project is based on the open-source Teapot dome oil field dataset.

Teapot Dome, located in the Powder River Basin, Wyoming, is a geologically significant field known for its complex structural features and productive hydrocarbon reservoirs. The field primarily produces from the Cretaceous and Pennsylvanian formations, with key reservoirs including the Tensleep Sandstone and the Frontier Formation.

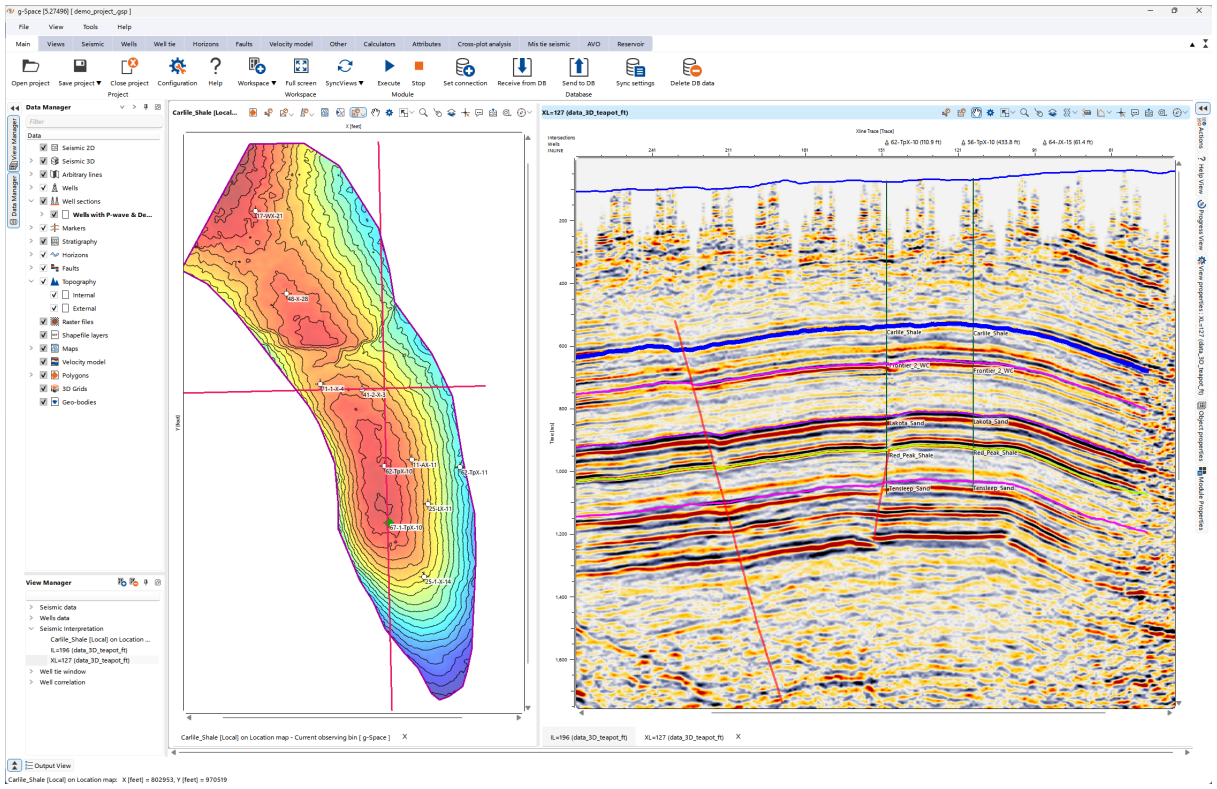
Dataset includes:

- 3D/2D seismic data
- Interpreted horizons
- Well data, including heads, inclinometry, logs, checkshots and markers

Please note that all data in the Demo Project is presented in the imperial system of units, with feet as the base unit of measurement. Project coordinate system is 32056 NAD27 / Wyoming East Central.

© 2026 Geomage 2003 LTD

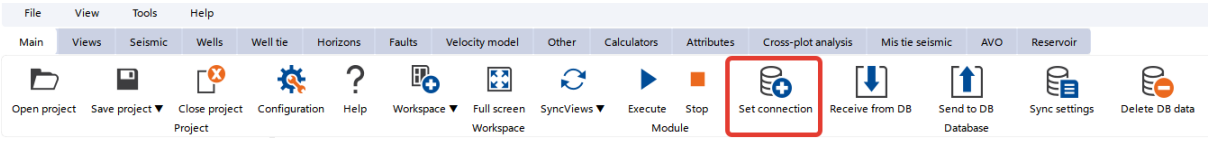
Several workspaces have been created in the demo project to help you familiarize yourself with the g-Space interface for different workflows, such as seismic interpretation, well correlation, and well tie.



4.7 Remote database

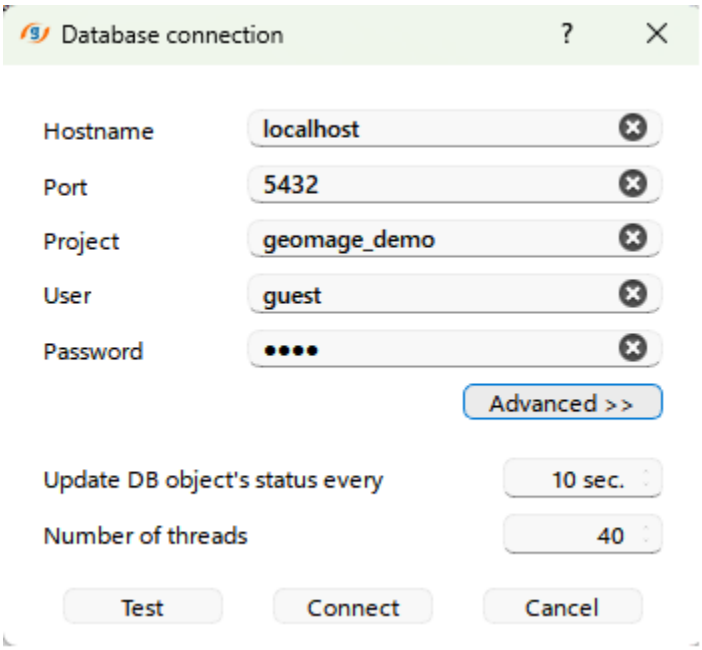
g-Space allows you to work with a remote database and offers a variety of tools for multi-user mode.

To set up a connection with the remote database, select "Set connection" in the main toolbar:



In the Connect to g-Space database window, enter the connection settings and click the "Connect" button:

© 2026 Geomage 2003 LTD



More information about working in multi-user mode you can find here: Multi-user mode 795.

© 2026 Geomage 2003 LTD

Import data

5 Import data

This chapter will cover the ways how to integrate different types of data in g-Space.

g-Space supports the following types of data:

- Seismic 373 - SEGY 2D and 3D data
- Well data 391 - well headers 393, inclinometry 397, markers 403, well logs 407, checkshots 414, comment logs 421
- Seismic horizons 438
- Fault sticks 441
- Shape files 443
- Polygons 444
- Topography 445
- Geo-referenced images 446
- Grid data 450

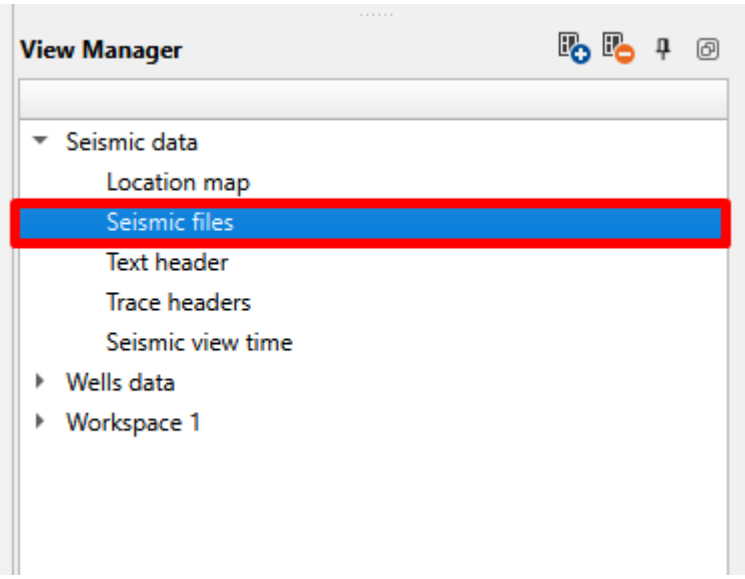
For more information about supporting formats please refer to chapter Formats 506

5.1 Import Seismic

Step 1

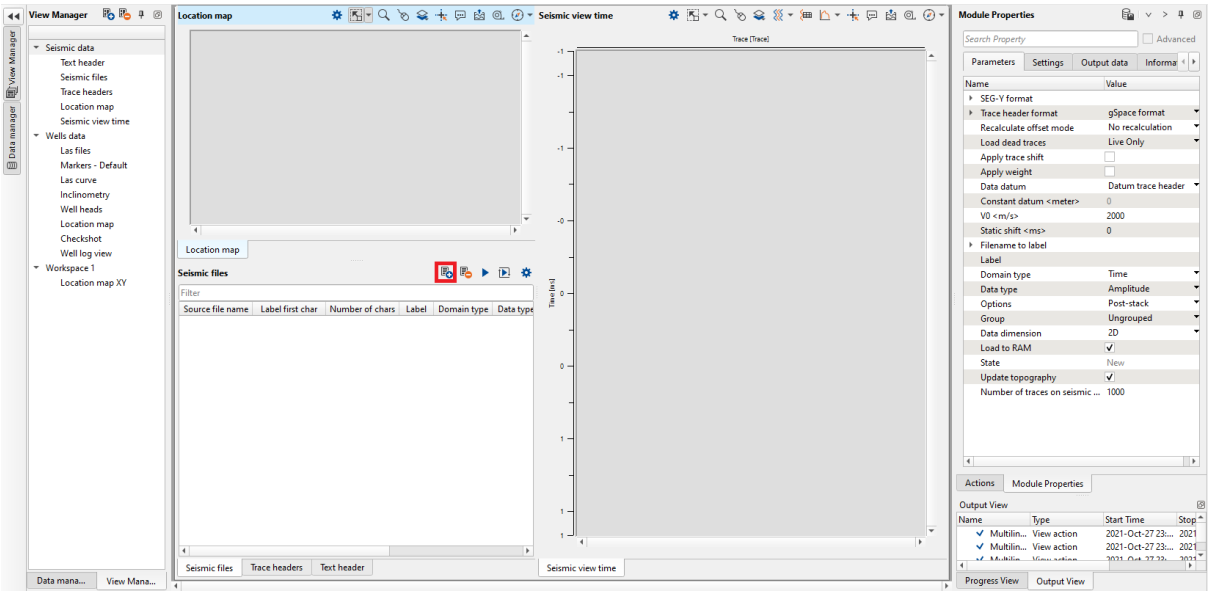
Go to the **seismic files** view of the seismic data 228 predefined workspace.

For this choose the **seismic files** view in the view manager 189:

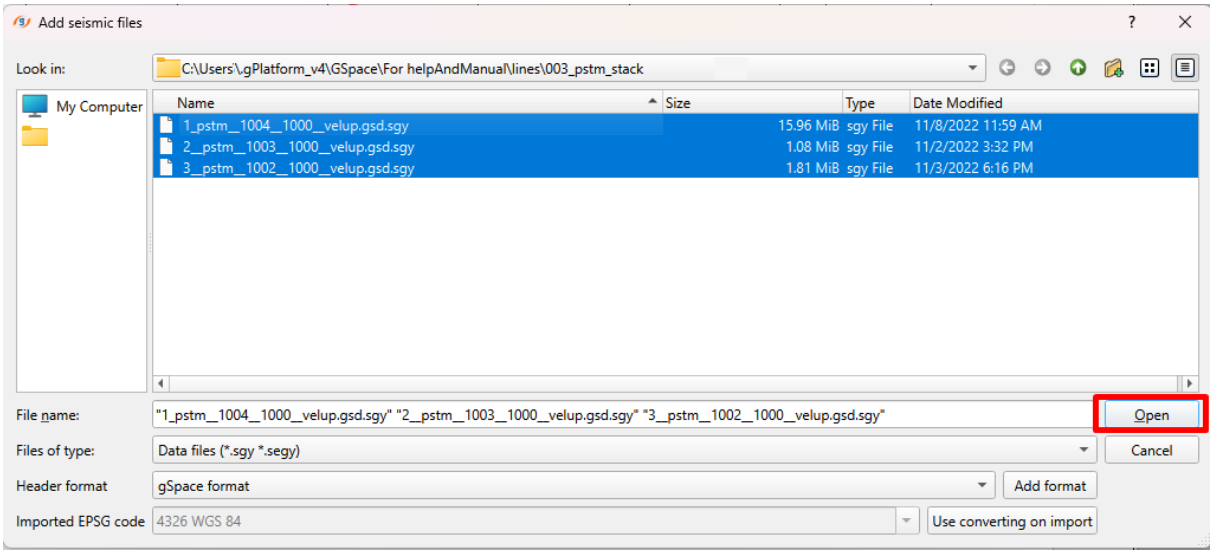


© 2026 Geomage 2003 LTD

In the **seismic files** view press **Add files** button:



Locate seismic data and press Open:

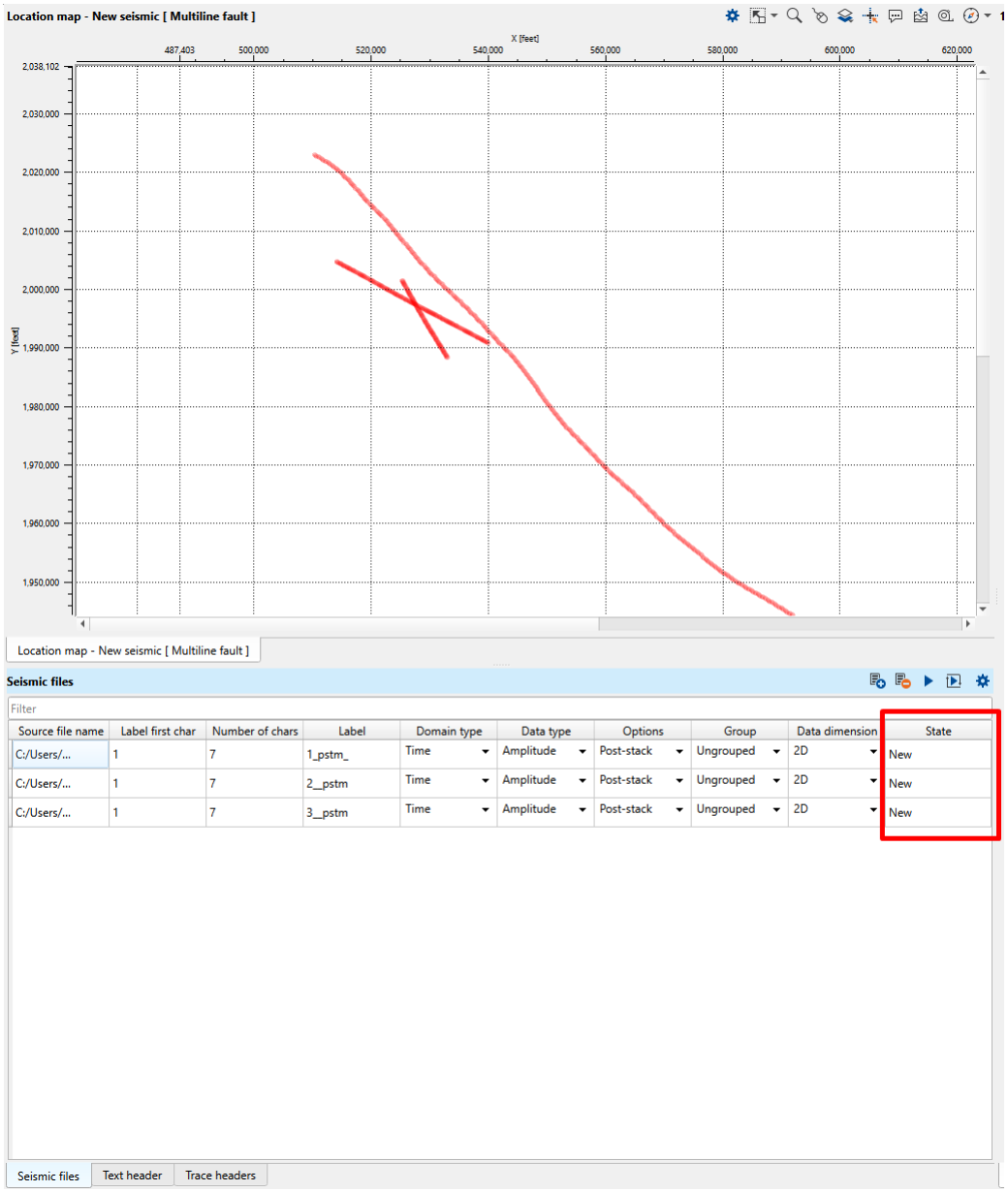


Step 2: Data Preview and setting parameters

In the **seismic files** view set Domain (time/depth), Data type (amplitude/velocity/attribute), Options (Pre-/Post-stack), add to group (for example processing type), Data dimension (3D/2D).

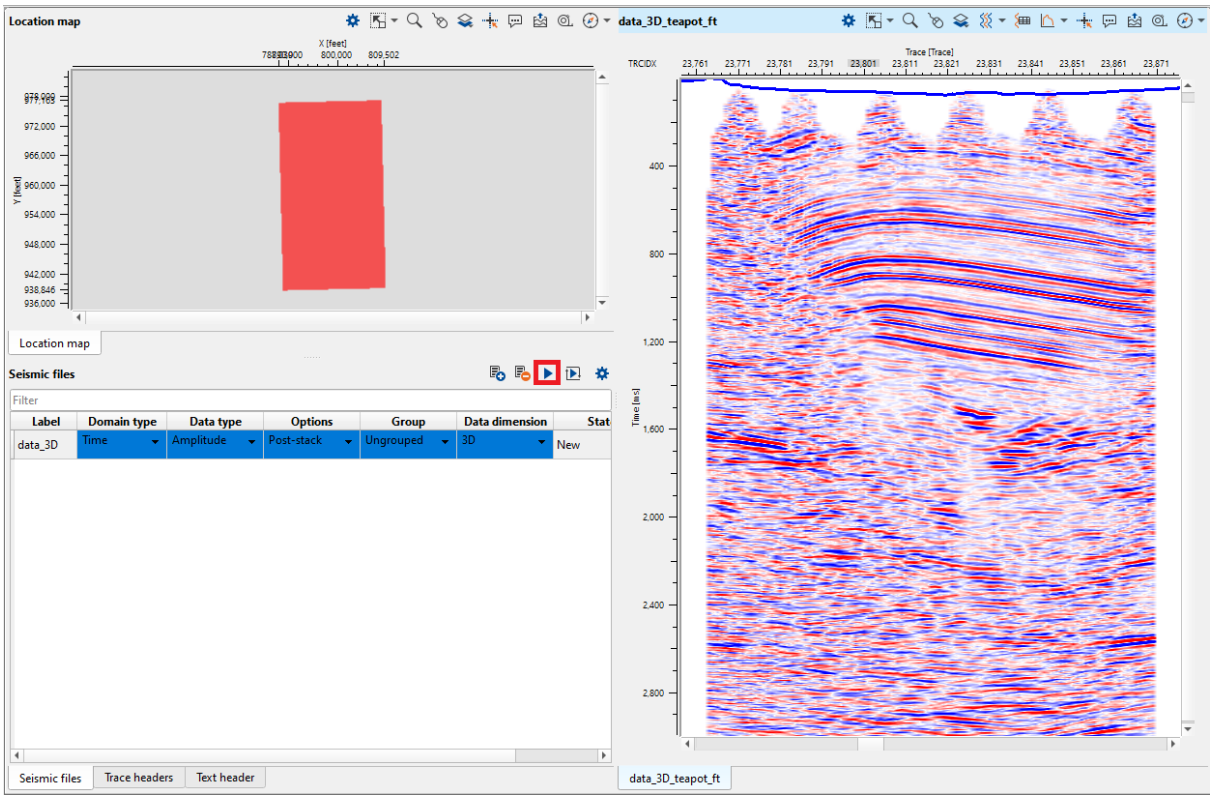
© 2026 Geomage 2003 LTD

On the **location map** the loading lines or bins will appear in **RED** - it means they haven't been loaded yet. Also they marked as 'New' in the **State** column in the **seismic files** view (may not be displayed if the default header format differs from the actual seismic):

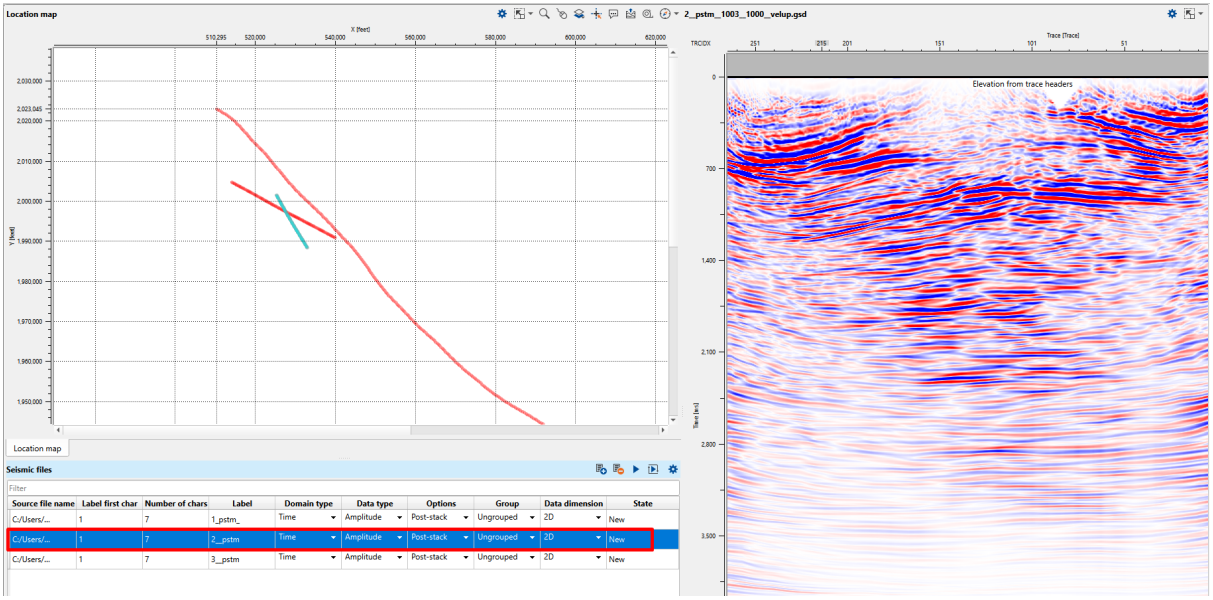


In case of 3D bins will appear in **RED**:

© 2026 Geomage 2003 LTD

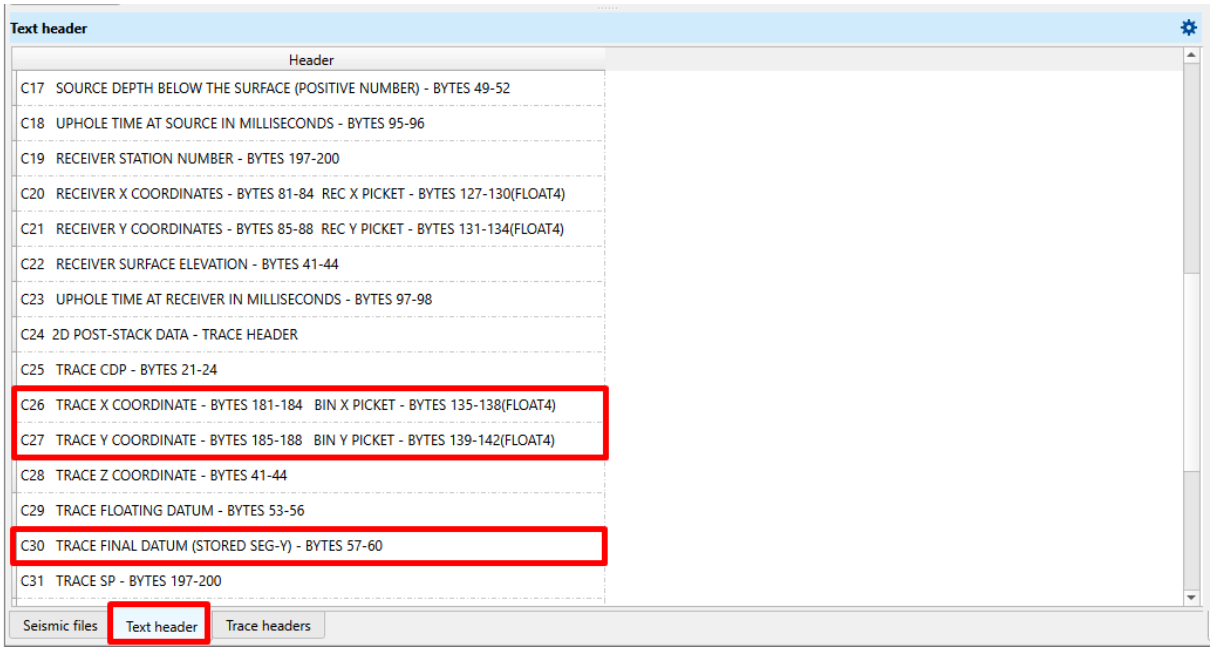


Choose specific file in the **seismic files** view to see the traces:

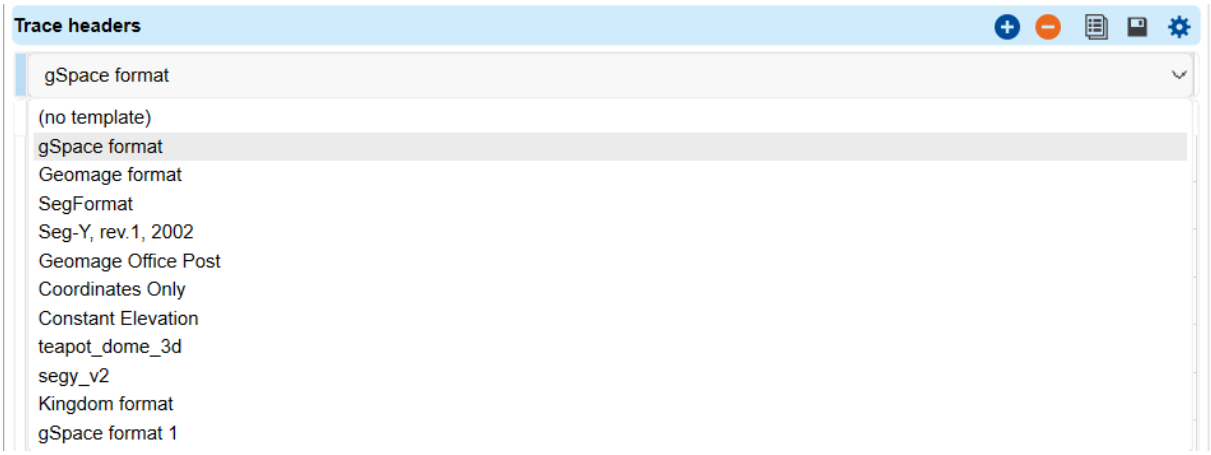


The text headers usually contain the main trace geometry information. In the **text header** view you can check which bytes contain it (X, Y and elevation in the case below):

© 2026 Geomage 2003 LTD

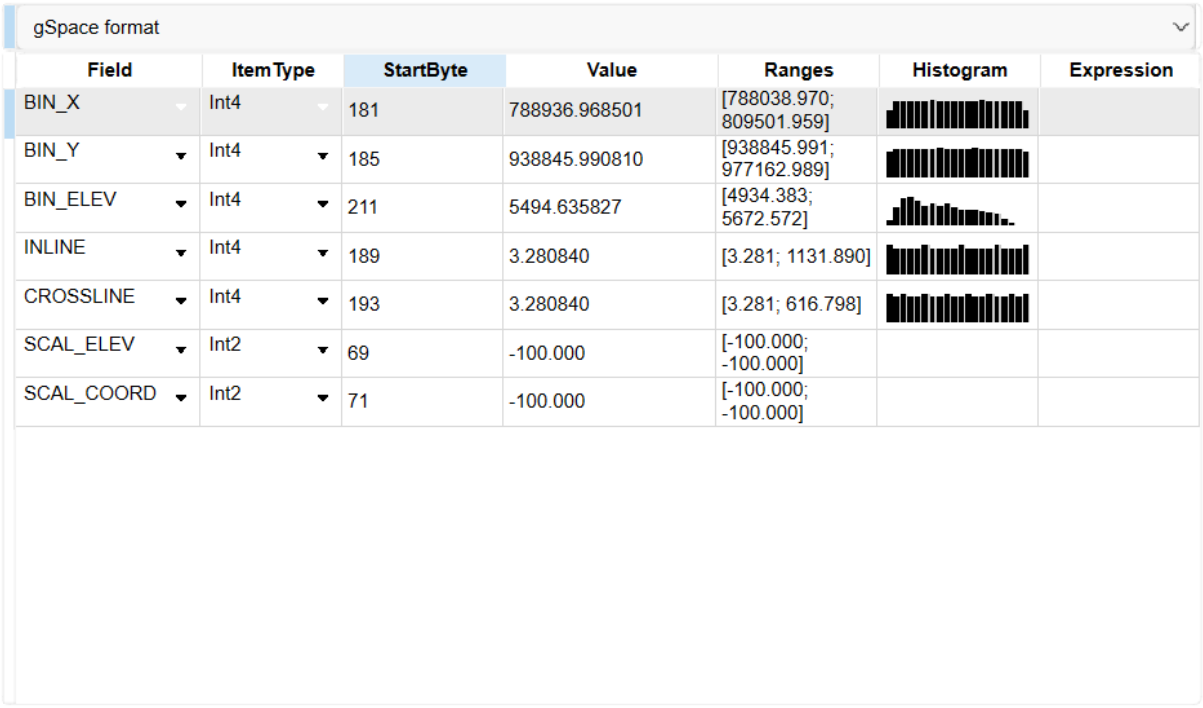


In the t**race headers** view choose one of the available trace header formats:

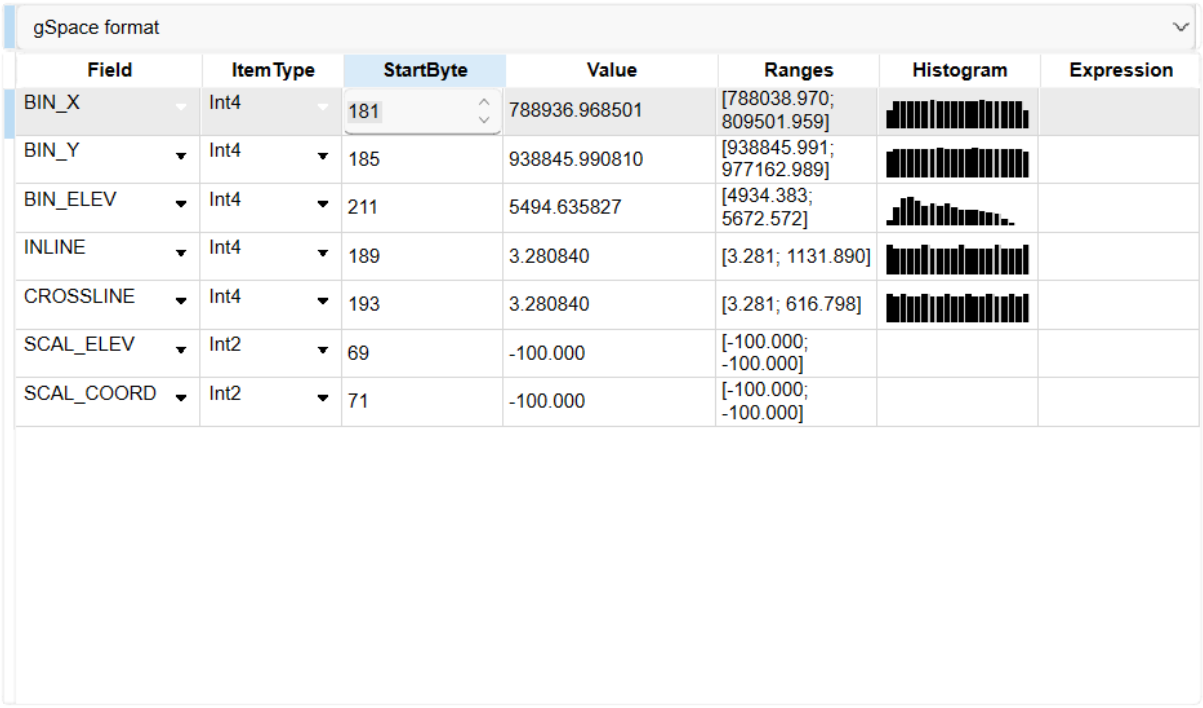


After that check that the bytes of the main fields are set correctly (trace X, Y and elevation and coordinates scale factors and other). To determine that you have your bytes set correctly the **Ranges** and **Histogram** columns can help you. The **Value**, **Ranges** and **Histogram** columns are calculated in the background: the table appears at once with the other columns, and these three are filled in when the calculation finishes, which can take a while for large data. Values already calculated for a file are kept, so selecting the same file again with the same trace header format shows them immediately.

© 2026 Geomage 2003 LTD

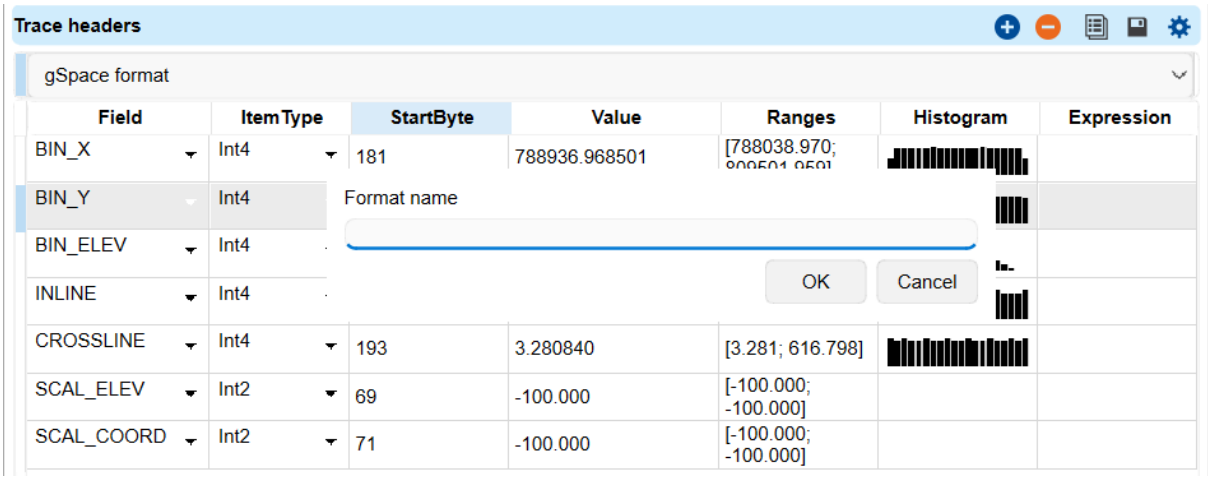


In case your seismic data in a different format than all available, select gSpace format and type **StartByte** value in any line according to your data:



After setting the new value, a window with a new format creation will appear:

© 2026 Geomage 2003 LTD

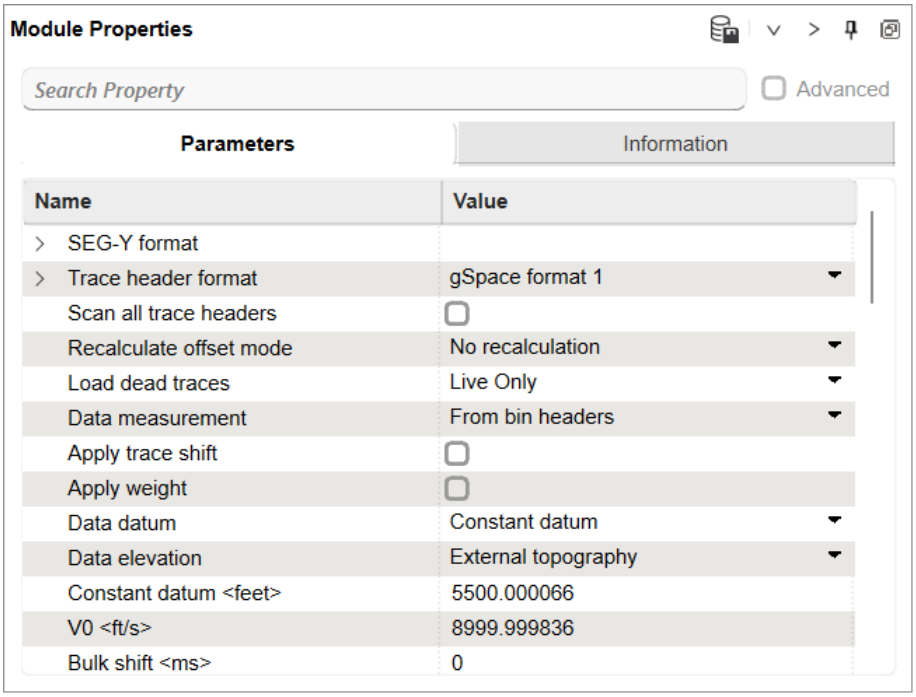


Type your format name and continue to change headers according to your data.

Once created, you can choose the new format later in other projects..

You have an option to use expression instead of trace headers. For this type an expression in the Expression column. To reference to the trace field use "data type (byte)". For example, the expression "int4(185)" returns the value read as a 4-byte integer from byte number 185. All available functions you can find here 521.

In Module properties 197 the additional loading parameters can be set such as Data Datum, Replacement velocity:



© 2026 Geomage 2003 LTD

The **Scan all trace headers** parameter is cleared by default. While it is cleared, a SEG-Y file imported with **Data dimension** 3D and **Options** Post-stack is read as a regular sorted bin grid derived from a few sampled trace headers instead of every trace header, which makes the import faster; if the sampled geometry is not a regular sorted cube, g-Space reads all trace headers automatically. Select **Scan all trace headers** to force the full per-trace header scan.

The seismic import parameters shown depend on the project Datum mode 368. In **Single datum** mode (the default, a simple SGY import) the per-file datum fields are hidden and the single project datum and replacement velocity are applied to every file. In **Multiple datum** mode the import view also exposes the **Data datum** (including **Constant datum**) and **V0** (replacement velocity) fields, so each seismic file can use its own datum and replacement velocity.

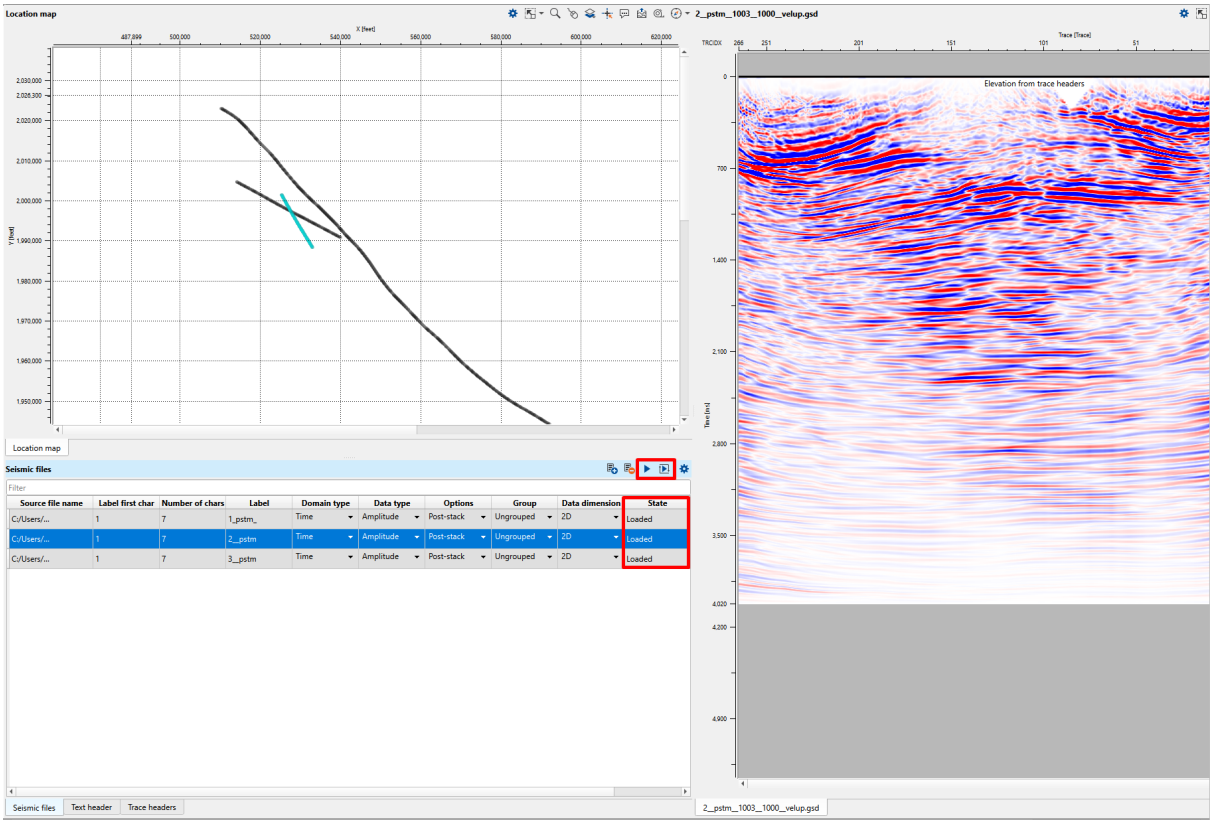
Note

g-Space uses topography information in some processes (such as velocity model creation). If you don't have topography information in your segy file you can set "Data elevation" parameter to "External topography" and it will be loaded from Geomage server.

Once all parameters are set press "Apply selected seismic" button or "Apply all seismic" button .

On the **location view** the loaded lines will appear in **Black** and **State** will change to "Loaded":

© 2026 Geomage 2003 LTD

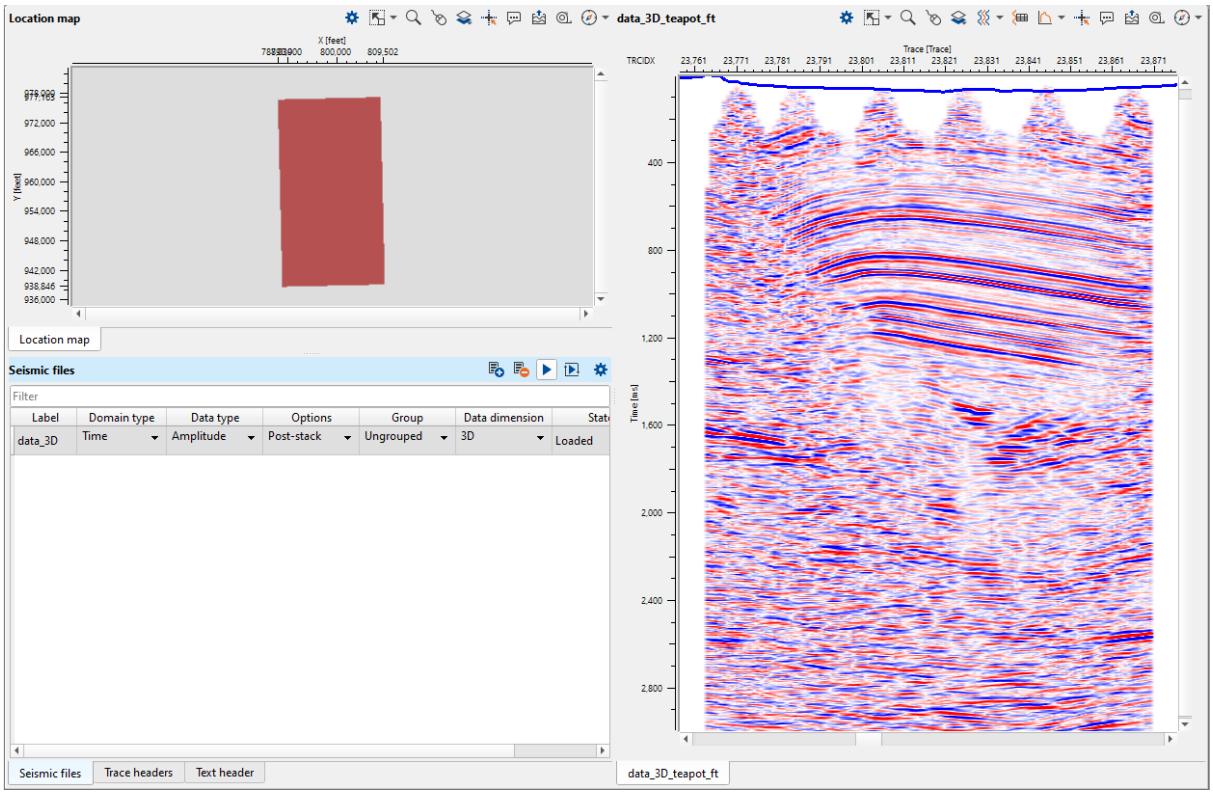


If you change the **Domain type** or the **Data type** of a file that is already loaded, the file is updated without being read again: its **State** stays "Loaded", the change is applied at once even for large data, and the Data Manager shows the seismic under the new domain and data type. Changing the **Options** value of a file that is already loaded and whose **Data type** is **Velocity** works the same way: switching between **RMS**, **Interval** and **Average** keeps its **State** "Loaded", whether you edit the **Options** cell of a single row or apply the value to several selected rows at once. The velocity type is stored as metadata, so the new value is applied at once even for large data, the procedures that select velocity volumes by type use it immediately, and the seismic stays available under its new velocity type in the Data Manager. For any other **Data type**, changing **Options** still returns the row to the "New" rows, so press **Apply selected seismic** or **Apply all seismic** once more.

If the seismic file itself has been moved to another location, you can point a row that is already loaded at its new path: click the **Source file name** cell of that row and select the same file in the dialog that opens (it lists *.sgy files). The row moves back to the "New" rows with the new path and **State** "New", so press **Apply selected seismic** or **Apply all seismic** once more to load it from there. If the newly selected file cannot be read with the parameters of that row, the row keeps its previous file and nothing changes.

In case of 3D on **location view** the loaded volume will appear in **Dark red**:

© 2026 Geomage 2003 LTD



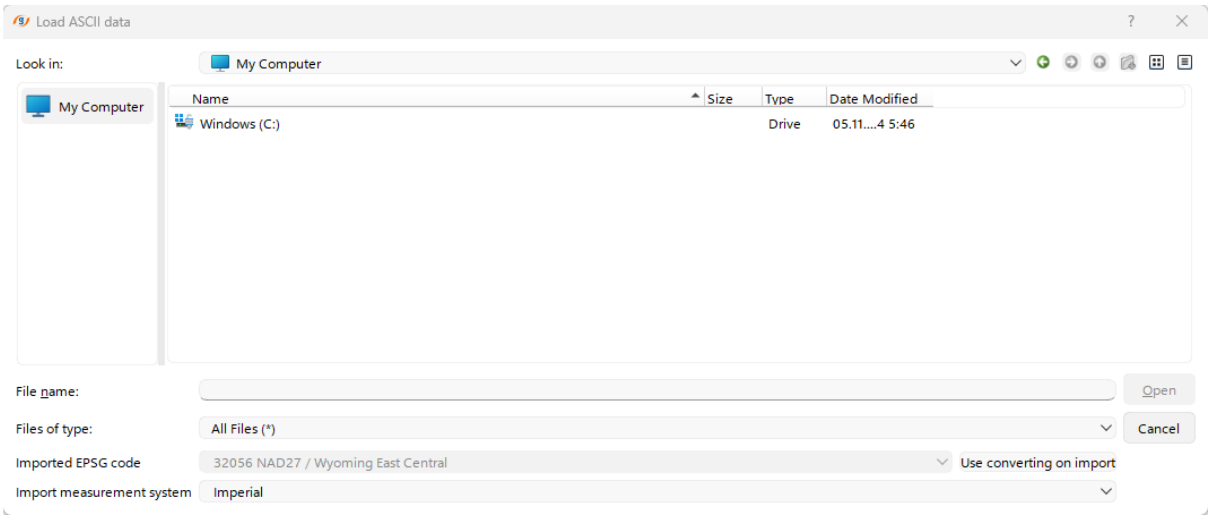
A 3D volume becomes the current volume of its survey as soon as it is applied. An Inline or Crossline view already open on that survey switches to the newly loaded volume, and its checkbox in the Data Manager 124 becomes available and checked for that view.
To display a different volume of the same survey, right-click it under **Seismic 3D** in the Data Manager and select **Set as current**.

5.1.1 Import attributes

To load data from an ASCII formatted file, navigate to the Seismic bar 44 and click **Import seismic data from ASCII**. The same import is also available from the Data manager 124 context menu of the seismic data node; see Convert ASCII to seismic 384 for the full procedure and its parameters.

A window will open; select your file, ensure that the correct coordinate system and units of measurement are selected, and then upload the file.

© 2026 Geomage 2003 LTD



5.1.2 Import ZGY

ZGY is a compressed 3D seismic volume format. g-Space imports ZGY `.zgy` files as 3D volumes (and 2D lines) through the seismic files view, the same place used to load SEG-Y.

Purpose

Load post-stack 3D seismic volumes stored in the ZGY format into a g-Space project. Geometry (inline / crossline, coordinates) is read from the ZGY metadata, so no trace-header mapping is required.

Launching the import

Open the **seismic files** view of the seismic data 228 workspace and press **Add files**.

Step-by-step

1. In the file dialog, set the file filter to **ZGY** and select one or more `.zgy` files.
2. Review the data preview: domain (time / depth), data type and dimension are detected from the file.
3. In Module properties 197 set additional loading parameters if needed (data datum, replacement velocity).

© 2026 Geomage 2003 LTD

4. Press **Apply selected seismic** to load the volume. Loaded data turns from red to black / dark red on the location map.

Notes

- ZGY carries its own geometry, so trace-header formats do not need to be configured as they do for SEG-Y.
- ZGY import is read-only; export back to ZGY is not provided. Export seismic as SEG-Y instead.

See Also

Import Seismic 373 | Seismic Data Formats 511 | Export seismic 494

5.1.3 Convert ASCII to seismic

Convert ASCII to seismic builds 2D lines and 3D volumes from a plain **ASCII** text file of scattered *X / Y / Z / Value* points (`.txt`). Unlike a raw trace load, this is an interpolation and gridding operation: the point values are resampled along Z and gridded into traces, and the result is stored internally as SEG-Y.

Purpose

Turn an unstructured table of point measurements into regular seismic traces. Each unique X / Y position becomes a trace, the *Z* column supplies the sample position (time or depth) and the *Value* column supplies the amplitude. Points are interpolated vertically to a fixed sample rate; for a volume the traces are then gridded into a full cube. The output is amplitude data, not geometry-only, and can be treated as any other seismic object afterwards.

Launching the import

The tool is reached from the Data manager 124 context menu on the seismic data node. Two entries are available:

- **Convert ASCII data to line** — produces one or more 2D lines.
- **Convert ASCII data to volume** — produces a 3D volume (pseudo-cube).

Either entry first opens a **Load ASCII data** file dialog where one or more text files are selected, then the **Import ASCII data** column-mapping wizard.

© 2026 Geomage 2003 LTD

Step-by-step

1. Right-click the seismic data node in the Data manager and choose **Convert ASCII data to line** or **Convert ASCII data to volume**.
2. In the **Load ASCII data** dialog, select one or more `.txt` files.
3. In the **Import ASCII data** wizard, set the column separator and the first / last data rows, then map the file columns to the **Survey ID** (optional), **X**, **Y**, **Z** and **Value** fields.
4. Set the vertical and interpolation options (domain, sample rate, interpolation method); for a volume also set the bin step and gridding options described below.
5. Confirm the wizard. Points sharing a *Survey ID* are converted together, so a single file with several IDs yields several lines.

Parameters

Column mapping:

- **Survey ID** — optional. When present, its value groups points into separate lines / surveys.
- **X**, **Y** — required. Trace position in the survey coordinate system.
- **Z** — required. Vertical sample position (time or depth).
- **Value** — required. Amplitude written into the trace.

Vertical / interpolation options (both lines and volumes):

- **Data is 2D** — chooses line versus volume output. It is preset by the menu entry used and is only editable when launched without a fixed choice.
- **Data is in time domain** — marks the output as time (otherwise depth).
- **Invert Z axis** — negates the Z column, for files where Z increases upward.
- **Output sample rate** — vertical sampling interval of the created traces.
- **Interpolation method** — how values are interpolated between points along a trace: **Linear** or **Spline**.

Volume gridding options (used only for Convert ASCII data to volume):

- **Step X**, **Step Y** — bin grid cell size used to build the output cube.
- **Kriging covariance** — covariance model used when filling empty bins: **Spherical**, **Gaussian** or **Exponential**.
- **Kriging search radius** — distance around each bin within which input traces are collected.
- **Kriging min number of points** — minimum neighbour count; if the radius yields fewer, the nearest points are used instead.

The converted data is held internally in the g-Space SEG-Y representation, so once imported it behaves like any other seismic line or volume.

© 2026 Geomage 2003 LTD

See Also

Import Seismic 373 | Seismic Data Formats 511

5.1.4 Import mistie cross-points

The **mis-tie cross-points table** is a line-intersection misfit table that records, for every point where two 2D seismic lines cross, the measured time misfit together with the time shift and phase rotation to apply to each line. g-Space can import this table from a tab-separated ASCII `.txt` file so a previously computed or edited set of intersection corrections can be reloaded into the project.

Purpose

Importing the cross-points table restores the state of 2D seismic mis-tie correction: the per-intersection time misfits, the time shifts and phase rotations for each line, and the checked (QC) flag. The imported values feed the mis-tie correction so the shifts and rotations can be reviewed, redistributed, or applied to the data without recomputing every intersection.

Requirements

The 2D lines referenced in the file must already be loaded, because each row is matched to an existing intersection in the project. Rows are located by their *X* and *Y* coordinates to the nearest intersection; if no intersection lies close enough, or if the named lines do not pass through that point, the row is skipped.

Launching the import

Run the mis-tie correction on your 2D seismic lines, then use the **Import cross-points table** command. A standard file dialog opens filtered to **Text files (*.txt)**.

Step-by-step

1. Make sure the 2D seismic lines are loaded and their intersections are available in the mis-tie correction.
2. Invoke **Import cross-points table** and select the `.txt` file to load.

© 2026 Geomage 2003 LTD

3. g-Space reads the file, skips the header row, and matches each data row to the nearest existing line intersection.
4. For every matched intersection the time shifts, phase rotations, time misfit, and checked flag are applied; the lines are re-ordered automatically if the file lists them in the opposite order.
5. Review the updated intersection misfits and shifts; unmatched rows are reported as warnings and left out.

File format

The file is tab-separated ASCII with a single header line followed by one row per intersection. Each row has ten columns:

- **X**, **Y** - coordinates of the line-intersection point (used to match the row to an existing intersection).
- **Checked** - QC flag, `Y` or `N`.
- **Misfit** - measured time misfit at the intersection.
- **Line1**, **Line2** - names of the two intersecting 2D lines.
- **Shift1**, **Shift2** - time shifts applied to each line.
- **Rotation1**, **Rotation2** - phase rotations (in degrees) applied to each line.

A row is ignored unless it has exactly these ten tab-separated columns, so keep the layout produced by the export intact.

See Also

Export mistie cross-points 491 | Seismic Data Formats 511

5.1.5 Import mute picking

Mute picking defines the top and bottom mute curves applied to pre-stack gathers. g-Space stores each mute curve in a `.corr` file, and this topic covers importing those files so that previously picked top or bottom mutes can be reused on a pre-stack dataset.

Purpose

Load top or bottom mute curves for pre-stack gathers from disk. Each imported curve is attached to a gather geometry as its top mute or bottom mute, so it can be reviewed on the gather view and used when the gathers are muted and stacked.

© 2026 Geomage 2003 LTD

Launching the import

Mute picking import works on a pre-stack (gather) dataset. In the Data manager 124, right-click a pre-stack seismic geometry and choose **Load mute picking**. The same action is available as **Load mute** from the pre-stack gather context, and it requires the geophysical interpretation license.

Step-by-step

1. Right-click the pre-stack geometry in the Data manager 124 and select **Load mute picking**.
2. In the **Load mute picking files** dialog, set the file filter to **Data Files (*.corr)** and select one `.corr` file (or several when loading in bulk).
3. Under **Picking type**, choose whether the file holds the **Top** or **Bottom** mute. The default is **Top**.
4. Press **Open** to load. The curve is attached to the target geometry as its top or bottom mute; when several files are loaded at once, each is matched to its geometry by the hash encoded in the file name.

Notes

- Top and bottom mutes are separate curves. Import the top and the bottom `.corr` files individually, choosing the matching **Picking type** for each.
- Mute curves are stored in the time domain of the gathers. The imported curve is interpolated across offsets when displayed and applied.
- When multiple files are selected, file names must contain the geometry hash written by the export (the `_mute_picking_` tag). Files that cannot be matched or read are listed in a "Files were not loaded" message.
- The action is available only with the geophysical interpretation license.

See Also

Export mute picking 493 | Seismic Data Formats 511 | Import seismic 373

© 2026 Geomage 2003 LTD

5.1.6 Import GSD

GSD (Geomage Seismic Data, `.gsd`) is Geomage's native seismic container. g-Space imports `.gsd` files through the seismic files view of the seismic data workspace, the same place used to load SEG-Y and `ZGY`. This topic covers importing GSD into a g-Space project.

Purpose

Load seismic stored in the Geomage GSD format into a g-Space project. A GSD file can carry 2D lines and 3D volumes, and it stores its own geometry, so no trace-header byte mapping has to be configured as it does for SEG-Y. GSD is import only.

Launching the import

Open the **seismic files** view of the seismic data 228 workspace and press **Add files** (also available from the right-click menu of the seismic files table).

Step-by-step

1. Press **Add files** to open the **Add seismic files** dialog.
2. Set the file filter to **Gsd data files (*.gsd)** and select one or more `.gsd` files. GSD is loaded exactly like SEG-Y and `.zgy` from this dialog.
3. Confirm the import settings (2D / 3D and pre-stack / post-stack). Because GSD carries its own geometry, these values are kept as chosen and no trace-header format needs to be set for the file.
4. The selected files are added as rows to the seismic files table. Press **Apply selected seismic** (or **Apply all seismic**) to load them. Loaded data appears on the location map.

Notes

- A single GSD file can hold 2D lines and 3D volumes; g-Space reads the geometry from the file, so no SEG-Y trace-header mapping is required.
- GSD import is read-only. There is no export back to GSD - export seismic as **SEG-Y** instead.

© 2026 Geomage 2003 LTD

See Also

Import Seismic 373 | Seismic Data Formats 511

5.1.7 Import other seismic formats

In addition to SEG-Y, `ZGY` and GSD 389, g-Space can read several other seismic formats. Most of them are field / raw formats that are **converted to SEG-Y** on import: **SEG-D**, **SEG-2**, **SEG-B** (SN338 / SN339), **MiniSEED** and **Seismic Unix** (`SU`). All of them are import only. This topic covers the shared import workflow.

Purpose

Bring seismic stored in vendor / field formats into a g-Space project. These are mostly pre-stack field or shot gathers and single-trace collections; on import they are converted to SEG-Y (or read into a gather), after which they are loaded like any other SEG-Y volume. The supported formats and the objects they carry are:

- **SEG-D** (`.sgd, .segd, .sd, .hd`) - pre-stack field / shot gathers. Converted to SEG-Y (Convert SEG-D to SEG-Y).
- **SEG-2** (`.dat, .seg2, .sg2`) - pre-stack field / shot gathers. Converted to SEG-Y (Convert SEG2 to SEG-Y).
- **SEG-B** (`.segb`), including the SN338 and SN339 variants - pre-stack field / shot gathers. Converted to SEG-Y (Convert SEG-B to SEG-Y).
- **MiniSEED** - traces / gathers. Read into a gather (Read MiniSeed).
- **Seismic Unix** (`.su`) - traces / gathers. Read into a gather (Load SU).

Launching the import

Each format has its own reader / converter procedure in the seismic data procedures. Start the procedure that matches your data - for example **Convert SEG-D to SEG-Y**, **Convert SEG2 to SEG-Y**, **Convert SEG-B to SEG-Y**, **Read MiniSeed** or **Load SU**.

Step-by-step

The SEG-D / SEG-2 / SEG-B converters share the same steps; MiniSEED and SU read a single file into a gather.

1. In the converter, add the field files to the **Input files** list, using the file filter that matches the format's extensions.

© 2026 Geomage 2003 LTD

2. Run **Read headers** to scan the files. The **Common Information** and **Traces information** tables preview the geometry and trace headers detected in each file.
3. Set the **Output file**: file type **SEG-Y**, the output file name, and the **Trace header format**. Set any format-specific options - for SEG-B pick the **File format** (SEG-B or SN339); for SEG-D pick the **SEG-D version**.
4. Execute the procedure. The field files are written into a single SEG-Y (MiniSEED and SU are read straight into a gather instead).
5. Load the resulting SEG-Y into the project through the **seismic files** view - **Add files** - the same way you load any other SEG-Y 373.

Notes

- Importing pre-stack gathers (SEG-D, SEG-2, SEG-B and pre-stack SU / MiniSEED data) requires the `GEOPHYSICAL_INTERPRETATION` license; without it the pre-stack option is not available.
- These formats are import only. Once data is in the project it is stored and re-exported as SEG-Y.
- The SEG-B reader supports the SN338 / SN339 variants and exposes an **Advanced** header-size option; the SU reader adds options such as trace-header read mode, offset recalculation and data datum.

See Also

Import Seismic 373 | Seismic Data Formats 511

5.2 Import Wells

In this chapter we present how to load any Well-related data into g-Space project database

Selection of Wells data workspace 232 on View manager 189 activates corresponding workspace that include several views required for QC of the import process for all types of well data.

Here is the list of the data that is possible to import into g-Space

- Create well 392
- Import well headers 393
- Import inclinometry 397
- Import markers 403
- Import markers structure 406
- Import LAS files 407

© 2026 Geomage 2003 LTD

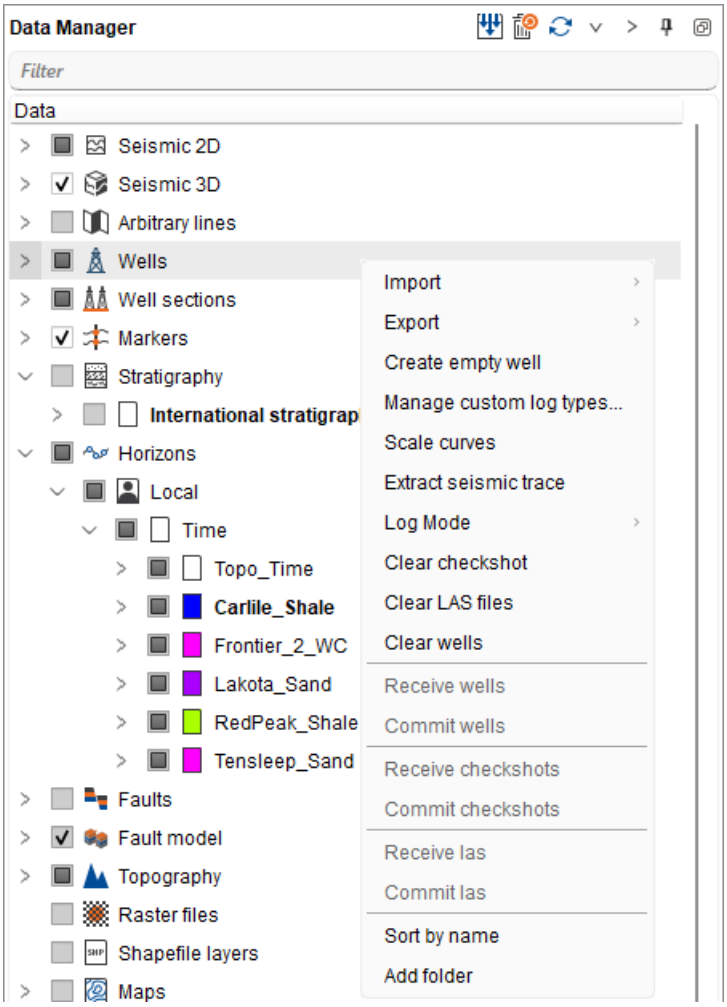
- Import checkshots 414
- Import comment log 421

5.2.1 Create well

The **Create empty well** wizard (class `GCreateEmptyWellWizard`) adds a new well record to the project when the user does not have a file to import. It is the right starting point when well geometry is known but the log and marker data have not been received yet, or when a synthetic / planning well is needed.

Launching the wizard

1. Open Data manager 124.
2. Right-click the **Wells** icon and choose **Create empty well**. The dialog that opens is titled **Create well**.

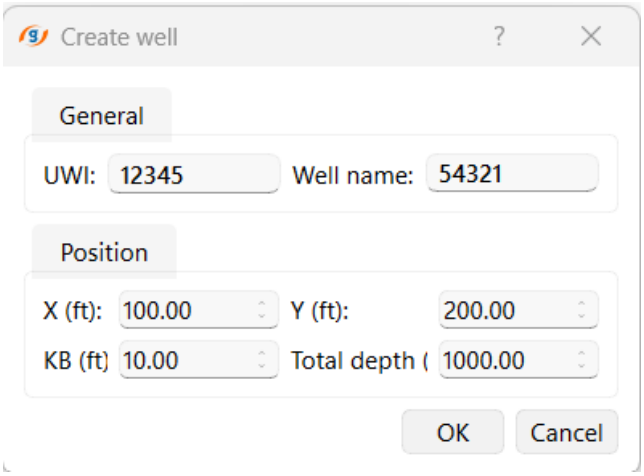


© 2026 Geomage 2003 LTD

Input Parameters

- **UWI** (unique well identifier) — required. The wizard checks the new UWI against all wells already in the project and refuses to accept duplicates.
- **Well name** — required.
- **X, Y** — surface location in the project's measurement system.
- **KB** (Kelly Bushing elevation) — reference elevation used to compute MD/TVD.
- **Total depth** — final depth of the wellbore.

If either UWI or name is left blank, or a duplicate UWI is entered, the wizard highlights the offending field and blocks submission.



After creation

The new well appears immediately in the Data manager 124. Well logs, inclinometry, checkshots and markers can then be attached via the usual import wizards (for example Import LAS files 407 or Import inclinometry 397).

See Also

- Import Wells 391
- Import LAS files 407
- Data manager 124

5.2.2 Import well headers

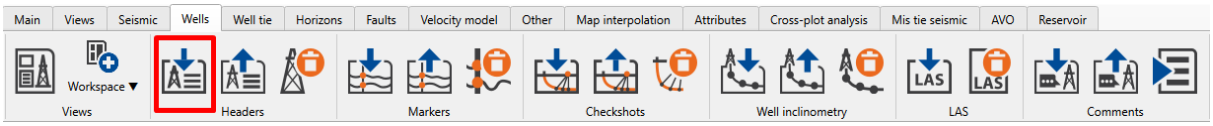
Import well headers steps:

© 2026 Geomage 2003 LTD

Step 1: Locating the file

Go to Ribbon 33 -> Wells bar 46 and click **Import wells headers from text file**.

For IHS 297 fixed or Petrel files, right-click the **Wells** folder in Data Manager 124, open **Import**, and choose **Well headers from IHS 297 fixed files** or **Well headers from Petrel**.



In the appeared dialog box locate the file and click open:

Step 2: Setting up reading parameters

g-Space will open file preview dialog were you can set the loading parameters such as: Separator type, Headers row, Start row, Date format.

In the **Original file** tab you will see how the file looks like and adjust loading parameters accordingly:

Step 3: Checking reading parameters

Click on Parsed file tab to check if you set reading parameters correctly and also you can decide which columns you want to add or skip:

Under **Fields connections**, use each drop-down to map a source column or choose **Skip**.

The well **Type** and **Status** are handled as two separate field connections. If your source file already contains columns named **Type** and **Status**, g-Space matches them to these fields automatically while loading the data.

The well **Type** and **Status** are handled as two separate field connections. If your source file already contains columns named **Type** and **Status**, g-Space matches them to these fields automatically while loading the data.

Note

If some of the fields are colored RED, it means that data doesn't have required type or reading parameters were set incorrectly:

© 2026 Geomage 2003 LTD

Step 4: Adding columns (Optional)

If you want to add columns which is not in the field click on the column, then Right Click Mouse on the column and choose the (Additional) column check box. There is an option to Rename columns:

Note

To load well headers in g-Space the file must provide these fields: ***UWI, X (Easting) and Y (Northing)****.*

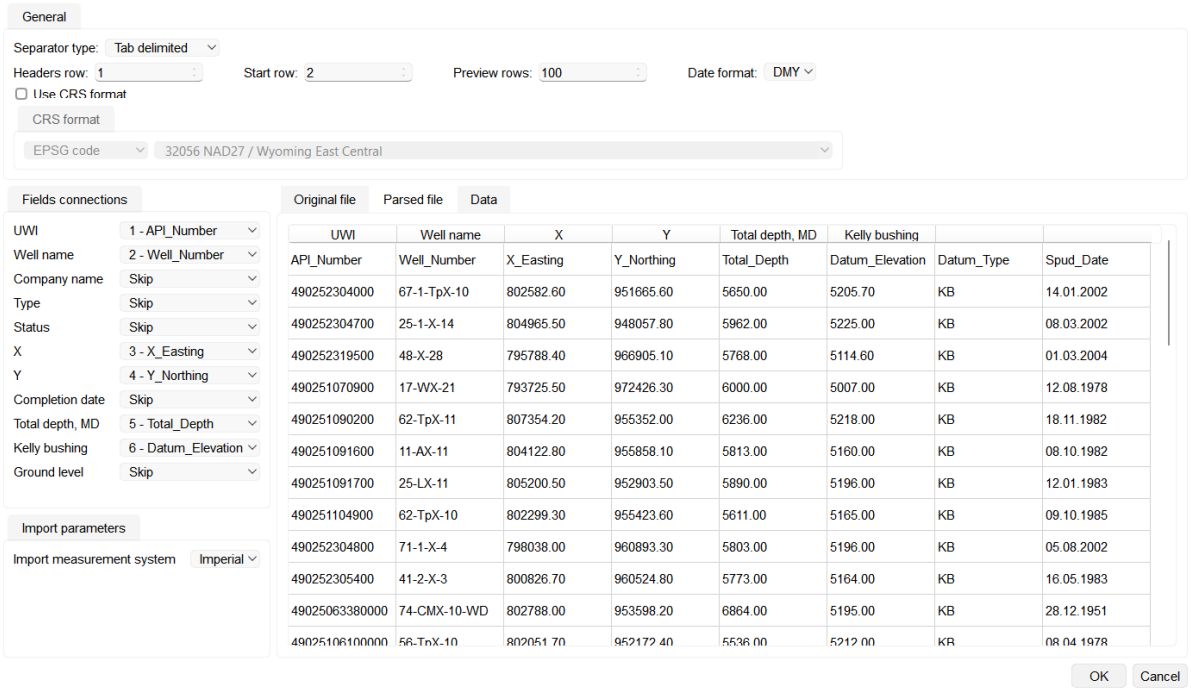
Additional attributes that can be matched and loaded:

- Total depth, MD (measured depth)
- Well type
- Well status
- Ground level (GL)
- Company name
- Completion date

Step 5. Checking the loading settings

In Data tab you can see if all settings were done correctly and all necessary columns were included. After that click OK:

© 2026 Geomage 2003 LTD



Step 6: Controlling loaded data

To review imported well headers, open the Wells data workspace 232 and click a well on the Location map. The **Well heads** table displays the imported header data.

For more information about checkshot format please refer to Well Data Formats 507

Importing well headers from Petrel format

Besides plain text and CSV files, g-Space can read well headers directly from a Petrel well head export. In Data Manager 124, right-click the **Wells** folder, open **Import**, and choose **Well headers from Petrel**.

In the file dialog select one or more **Petrel well head files** (*.txt, *.dat) and set the coordinate reference system (CRS) and measurement system for the imported data.

A Petrel well head file declares its columns inside a **BEGIN HEADER** ... **END HEADER** block, with one column name per line; the well data starts on the line after **END HEADER**. g-Space recognizes the standard Petrel columns and maps them to well header fields:

- Name – well name
- UWI – unique well identifier
- Surface X, Surface Y – well head coordinates (required)
- Well datum name, Well datum value – Kelly bushing (KB)

© 2026 Geomage 2003 LTD

- TD (MD) – total depth
- Spud date – completion date
- Operator – company name

Note

Surface X and Surface Y are mandatory. Any additional columns found in the header block are imported as extra well attributes.

5.2.3 Import inclinometry

Data format

Each inclinometry (deviation survey) file must provide three values per survey point:
- **MD** — measured depth along the wellbore.
- **INCL** — inclination, where 0 degrees is vertical.
- **AZIM** — azimuth, measured clockwise from north.

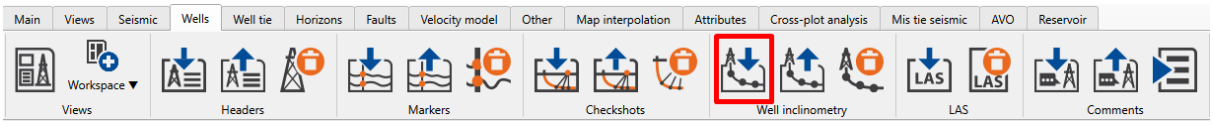
From these values g-Space calculates TVD, TVDSS, Easting (X) and Northing (Y) at each survey point.

Supported files are delimited text (`*.incl` or All Files; TAB- or comma-separated), Petrel deviation (`*.dev`) files, and well trajectory (`*.wt`) station files.

Import well deviation steps:

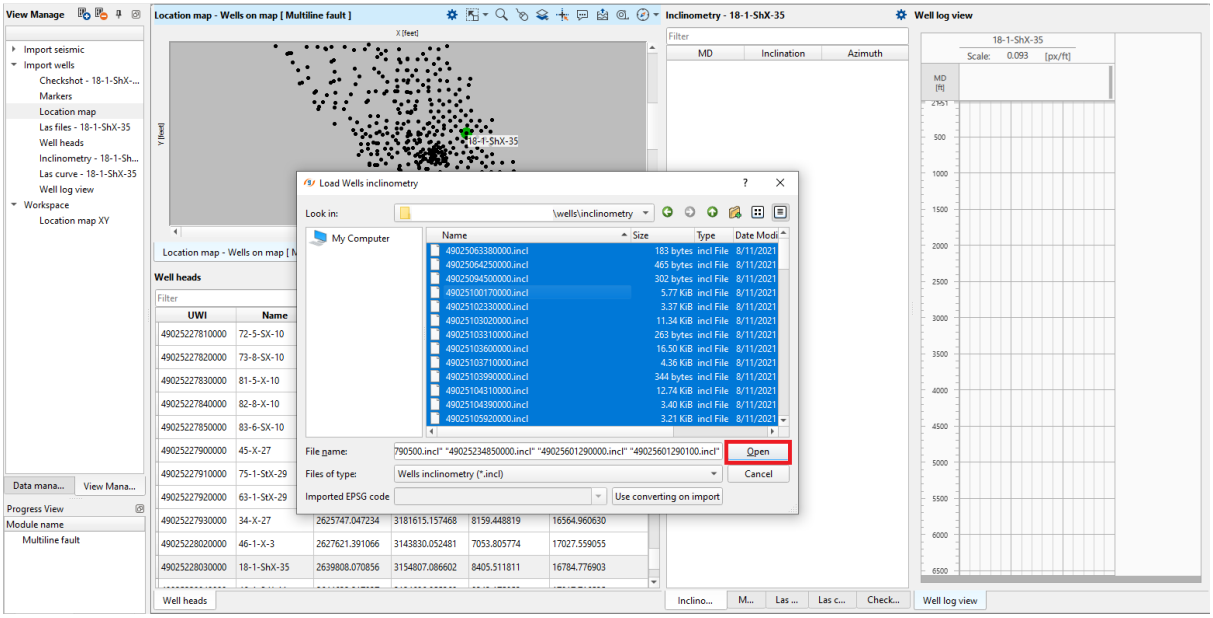
Step 1: Locating the file

Go to Ribbon 33 -> Wells bar 46 and click on Import wells inclinometry



In the appeared dialog box locate the files and click open:

© 2026 Geomage 2003 LTD

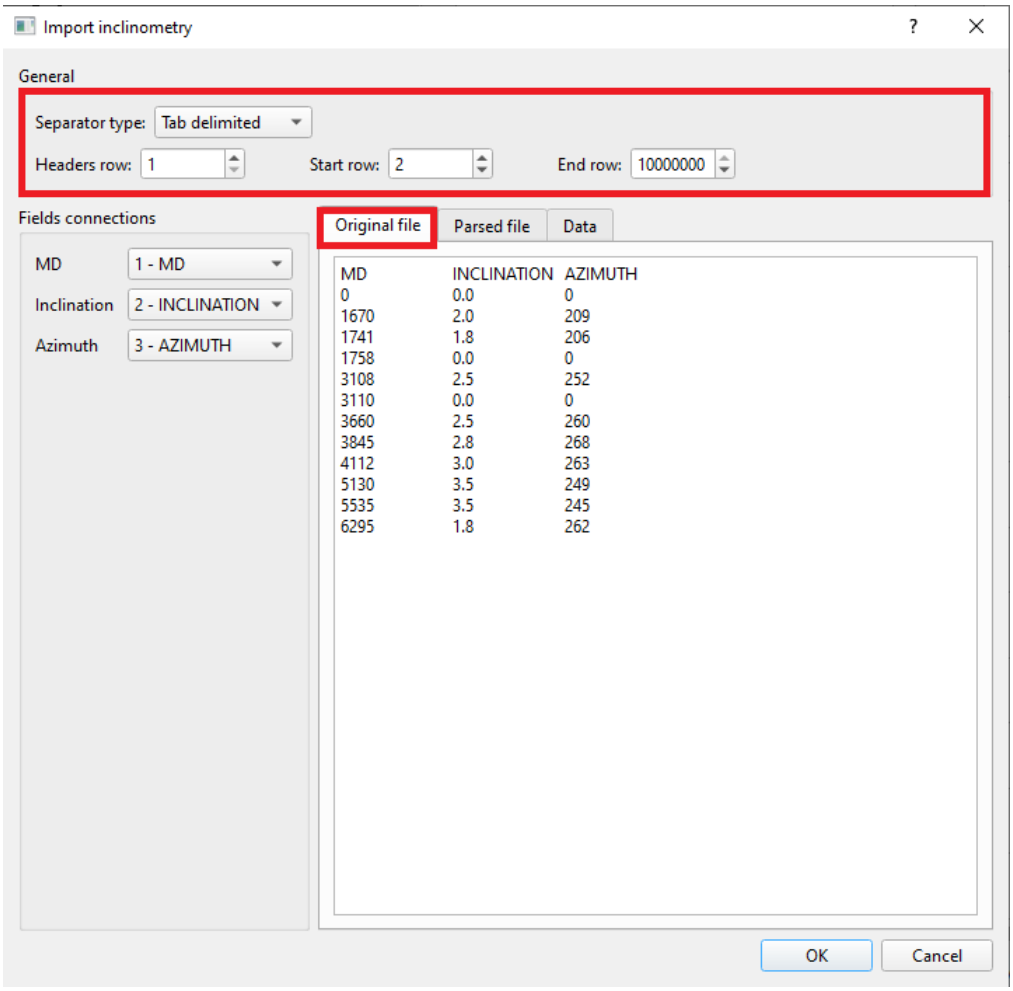


Step 2: Setting up reading parameters

g-Space will open file preview dialog were you can set the loading parameters such as: Separator type, Headers row, Start row.

In the **Original file** tab you will see how files looks like and adjust loading parameters accordingly

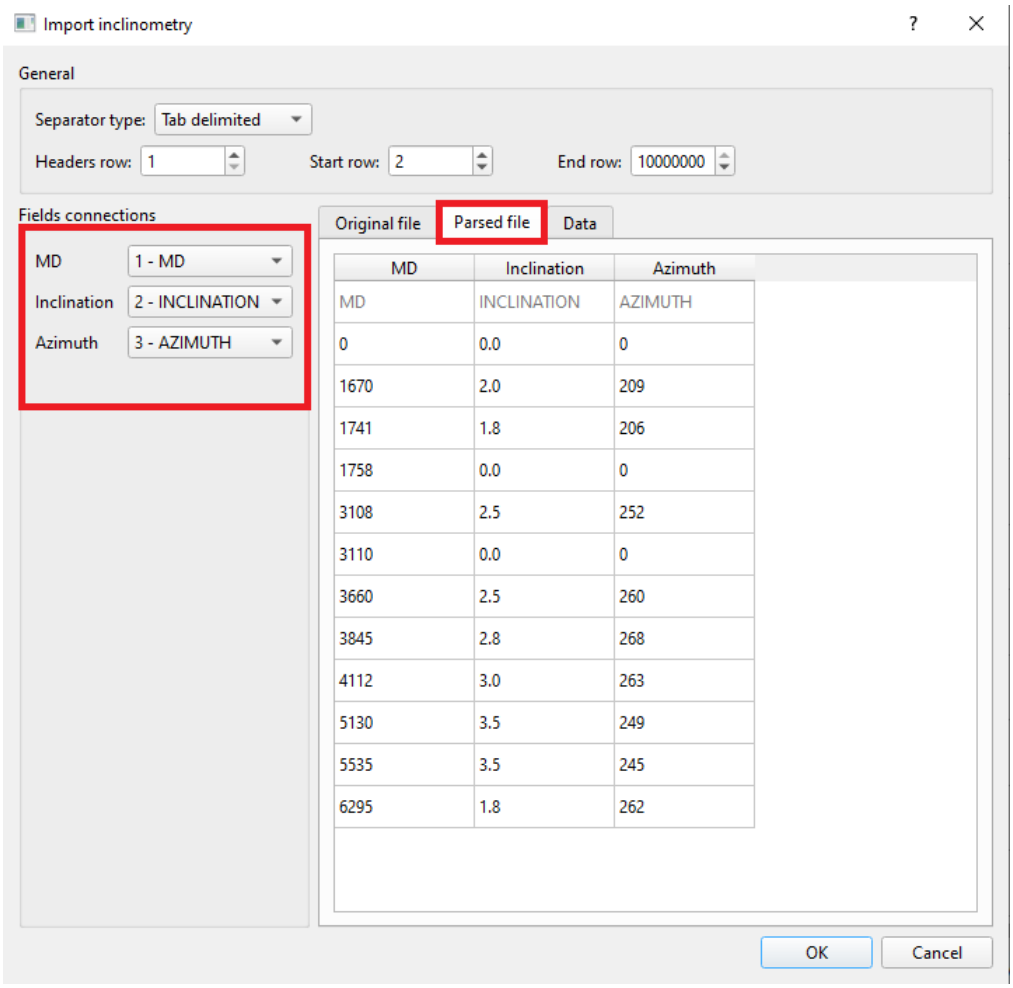
© 2026 Geomage 2003 LTD



Step 3: Checking reading parameters

Click on **Parsed file tab** to check if you set reading parameters correctly and also you can decide which columns you want to skip

© 2026 Geomage 2003 LTD

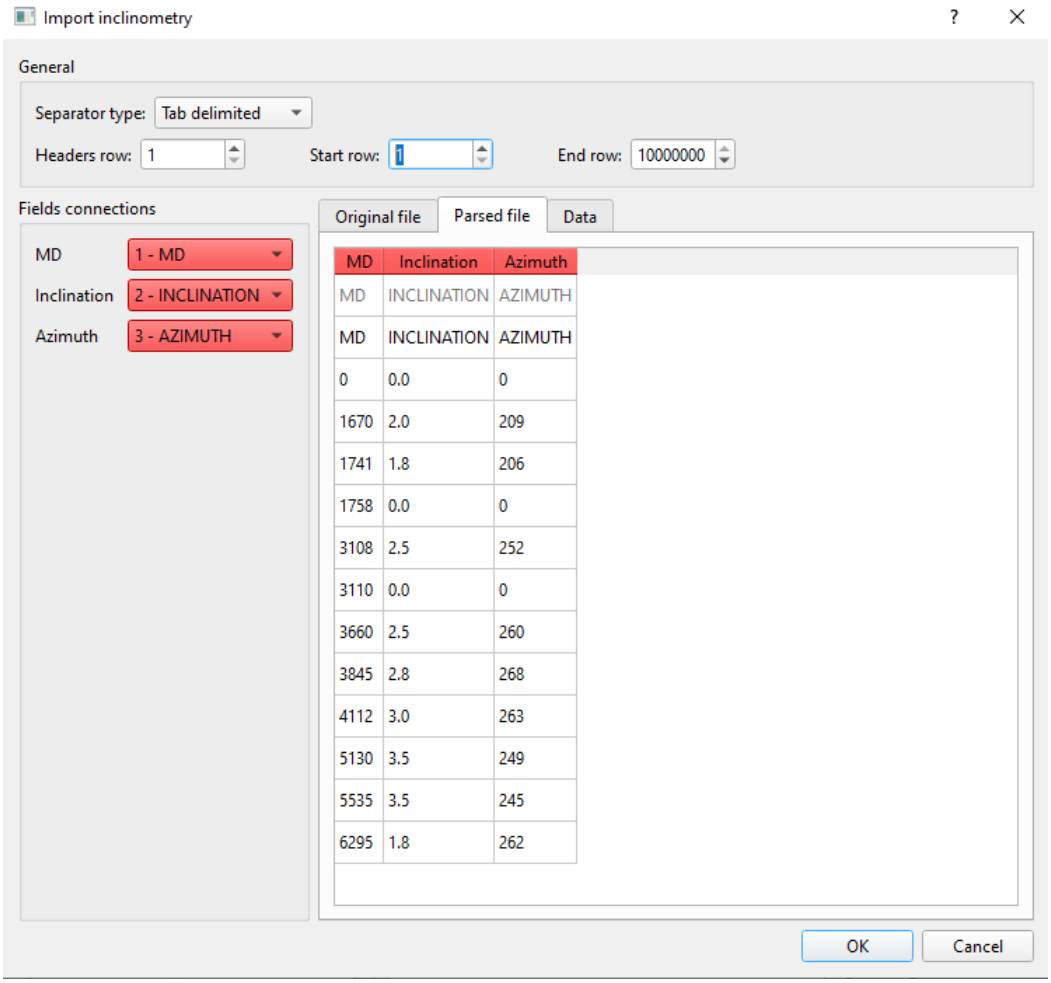


In the **Field connections** check the deviation parameters. If you have files with additional information you can chose skip unnecessary columns from drop-down box

Note

If some of the fields are colored RED, it means that data doesn't have required type or reading parameters were set incorrectly

© 2026 Geomage 2003 LTD

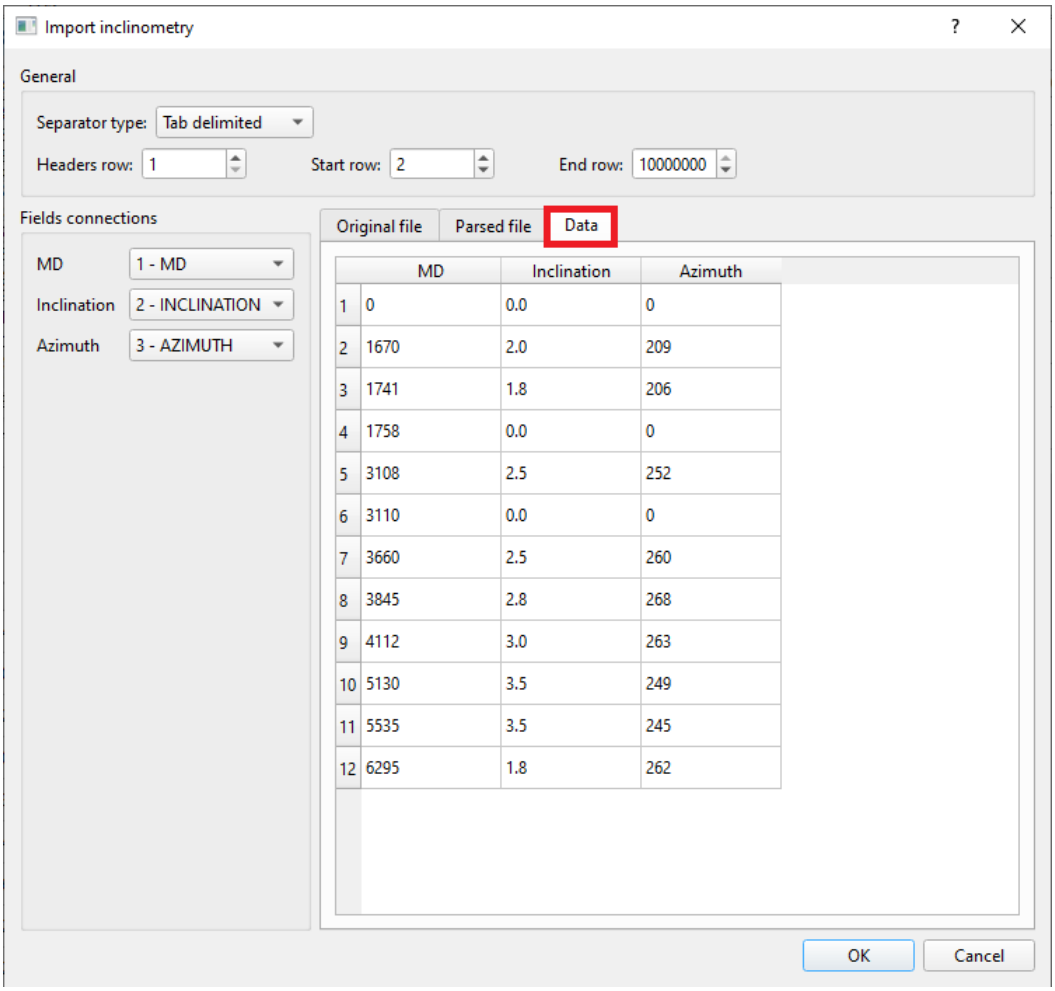


For more information about deviation format supporting by g-Space refer to Well Data Formats 507

Step 4. Checking the loading settings

In Data tab you can see if all settings were done correctly and all necessary columns were included. After that click OK

© 2026 Geomage 2003 LTD

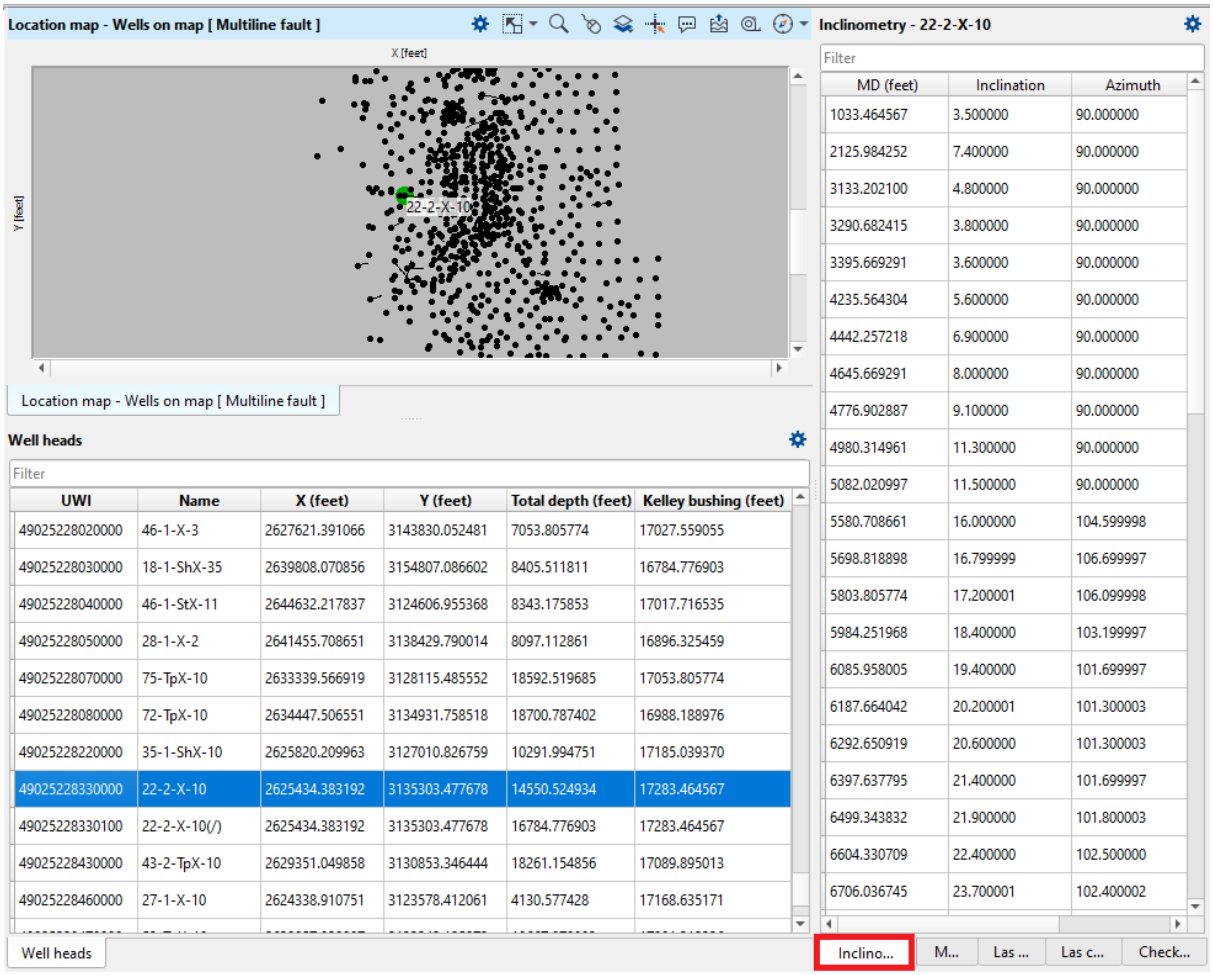


	MD	Inclination	Azimuth
1	0	0.0	0
2	1670	2.0	209
3	1741	1.8	206
4	1758	0.0	0
5	3108	2.5	252
6	3110	0.0	0
7	3660	2.5	260
8	3845	2.8	268
9	4112	3.0	263
10	5130	3.5	249
11	5535	3.5	245
12	6295	1.8	262

Step 5: Controlling loaded data

To check the data click on map view or in table below map - all loaded information will appear in the table **Inclinometry**

© 2026 Geomage 2003 LTD



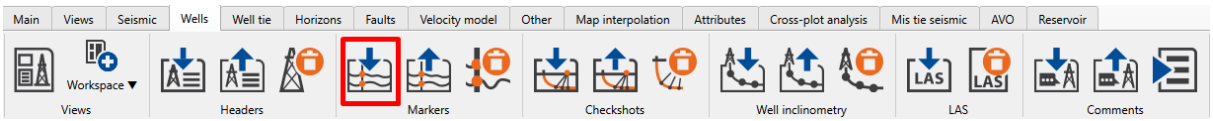
For more information about checkshot format please refer to Well Data Formats 507

5.2.4 Import markers

Import markers steps:

Step 1: Locating the file

Go to Ribbon 33 -> Wells bar 46 and click on **Import wells markers**



Accepted file formats: ***.marks2**, ***.txt** and ***.csv**.

There are two options to load markers:

© 2026 Geomage 2003 LTD

Option 1. Import well markers from MD format. Required columns:
- UWI or Well Name, Marker, MD

Option 2. Import well markers from XY format. Required columns:
- UWI or Well Name, Marker, X, Y, and either TVD or TVDSS

Age and **Type** can also be mapped when present in the file.

Petrel well-tops format

g-Space also reads well markers exported from **Petrel** in the well-tops text format. In such files the columns are declared inside a **BEGIN HEADER** ... **END HEADER** block, one column name per line, and the marker data starts on the line after **END HEADER**.

When you select a Petrel well-tops file, g-Space detects this layout automatically: it sets the column separator to space and moves the **Start row** to the first data line, so no manual adjustment is normally required.

The Petrel column names are matched to the g-Space marker fields automatically:
- **Surface** -> Marker
- **Well** -> Well Name
- **Geological age** -> Age
- **TVT** -> TVD
- **Z** -> TVDSS

Quoted marker or well names that contain spaces are kept as single values.

In the file dialog, select the marker file and click **Open**.

Step 2: Setting up reading parameters

g-Space opens the **Import markers** dialog. Under **Markers input format**, choose **MD** or **XY**, then review **Separator type**, **Headers row**, **Start row** and **Preview rows**.

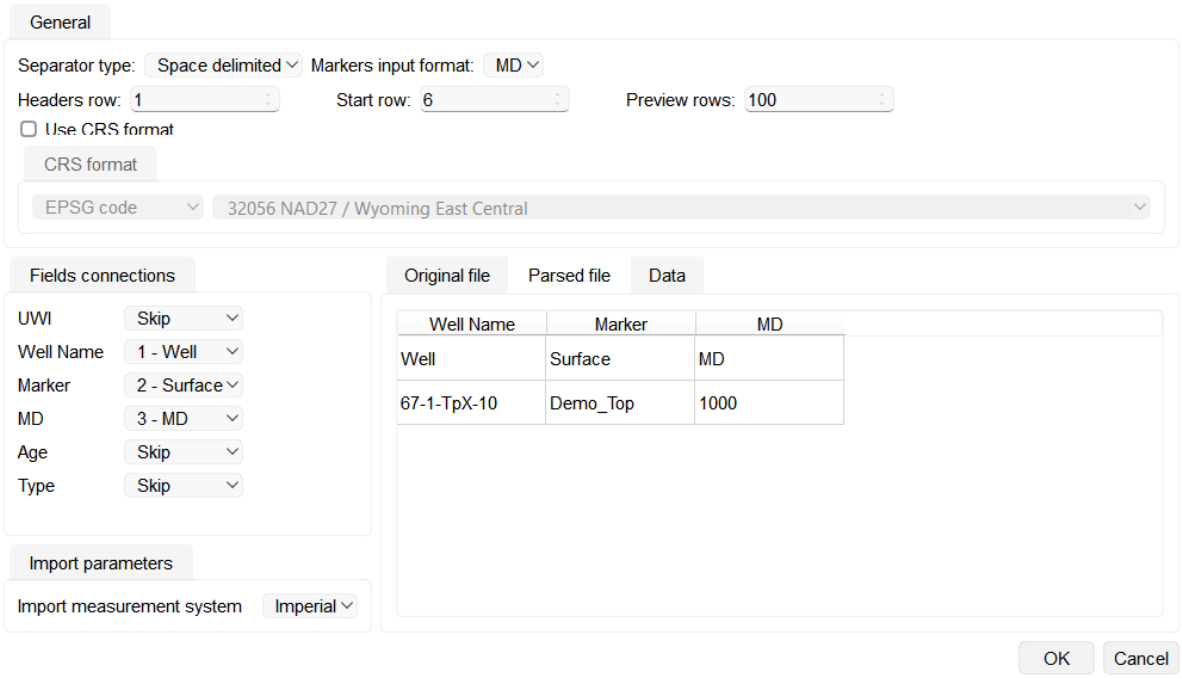
Use **Original file** to confirm the source layout. For ordinary text files, adjust the reading settings if necessary. Petrel well-tops files use the automatically detected settings described above.

Step 3: Checking reading parameters

Select **Parsed file** to confirm that the source rows are split into the expected columns.

© 2026 Geomage 2003 LTD

In **Fields connections**, verify **UWI** or **Well Name**, **Marker** and **MD** for MD input, or **UWI** or **Well Name**, **Marker**, **X**, **Y** and either **TVD** or **TVDSS** for XY input. Select **Skip** for unused optional columns.



A Petrel well-tops file after automatic detection. Well is mapped to Well Name, so UWI can remain set to Skip.

Note

If a required field is red, select a valid source column before continuing.

For more information about marker formats supported by g-Space, refer to Well Data Formats 507

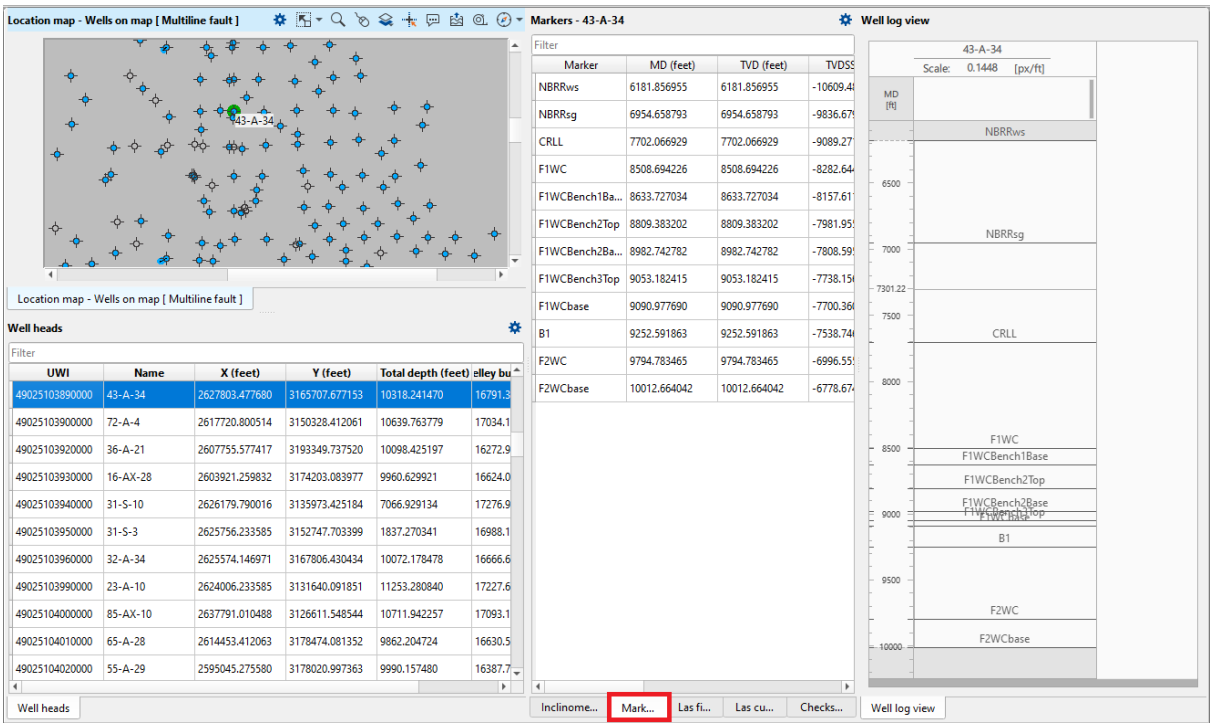
Step 4. Checking the loading settings

Select **Data** to review the mapped rows. When the values are correct, click **OK**.

Step 5: Controlling loaded data

To check the data click on map view or in table below map - all loaded information will appear in the table **Markers** and you can check them on **well log preview**

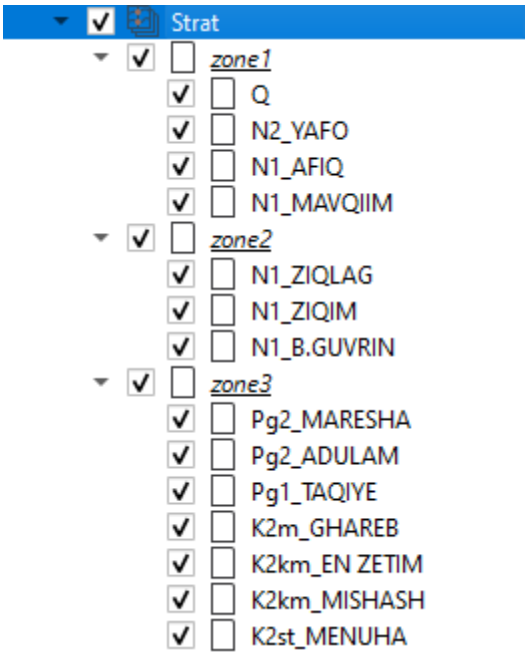
© 2026 Geomage 2003 LTD



For more information about markers format please refer to Well Data Formats 507

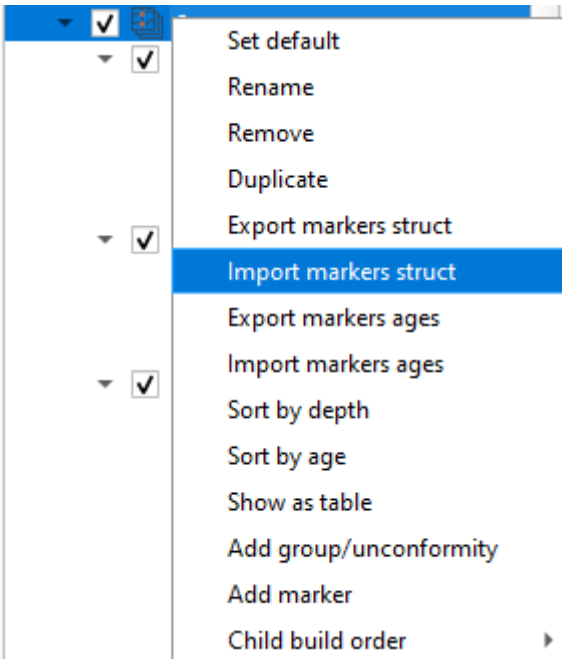
5.2.5 Import markers structure

Sometimes you need to organize you markers with different levels of hierarchy. In g-Space it is possible using marker groups:



© 2026 Geomage 2003 LTD

To import markers structure press Right Click Mouse on Markers set icon in Data Manager 124 and choose **Import markers struct**:



Locate the file with markers structure and it will apply automatically.

The markers structure is a g-Space binary **Marker set struct** file with the ***.msstruct** extension. It is created by g-Space when you choose **Export markers struct** on the same menu and stores the complete marker hierarchy.

The hierarchy can also be built and edited manually in the Data Manager 124 without importing a file: right-click the markers to create groups and drag the markers into them.

Tip

The hierarchy is preserved across export and re-import, so use consistent marker naming to keep the structure aligned between projects.

For more information about markers structure format please refer to Well Data Formats 507

5.2.6 Import LAS files

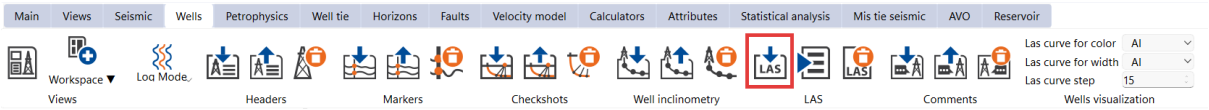
The **LOG** group of the Wells bar 46 provides two separate buttons for loading LAS files: **Load LAS files** for a quick, simple load, and **Log Loading Wizard** for an advanced load with full control over the loading process.

© 2026 Geomage 2003 LTD

Load LAS files (simple)

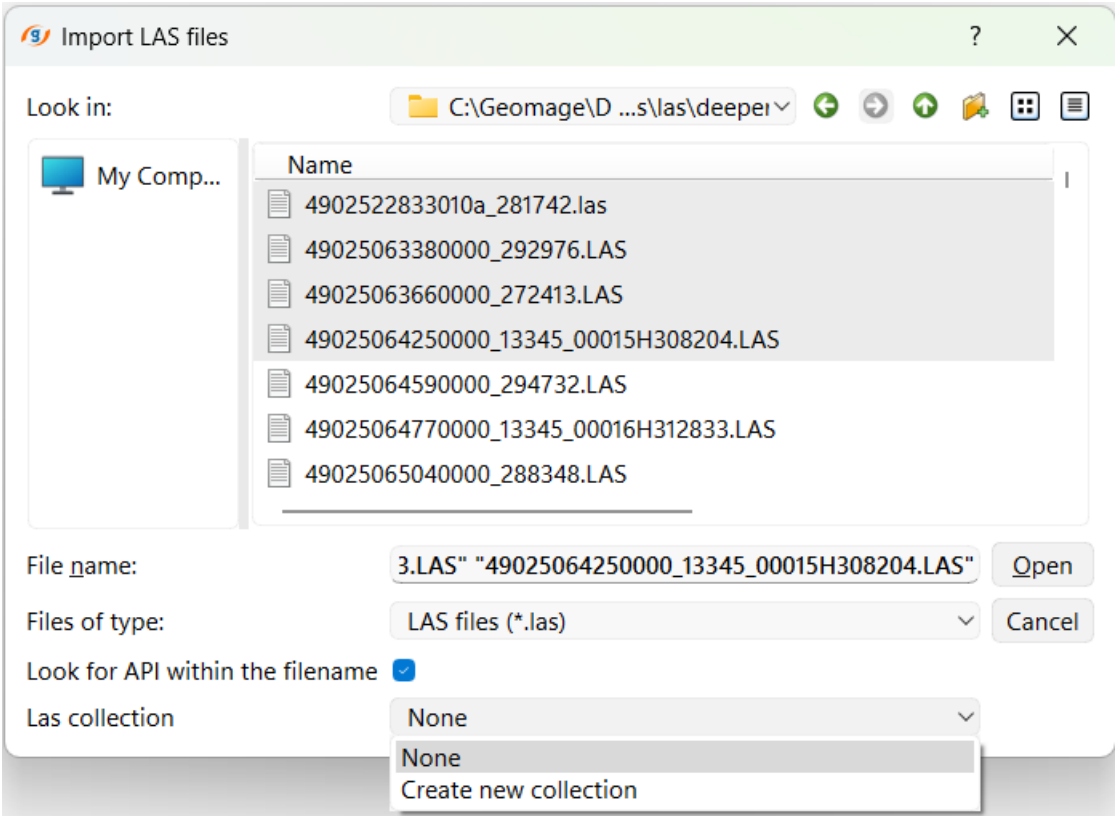
Step 1: Locate the file

Go to Ribbon 33 -> Wells bar 46 and click on Load LAS files button

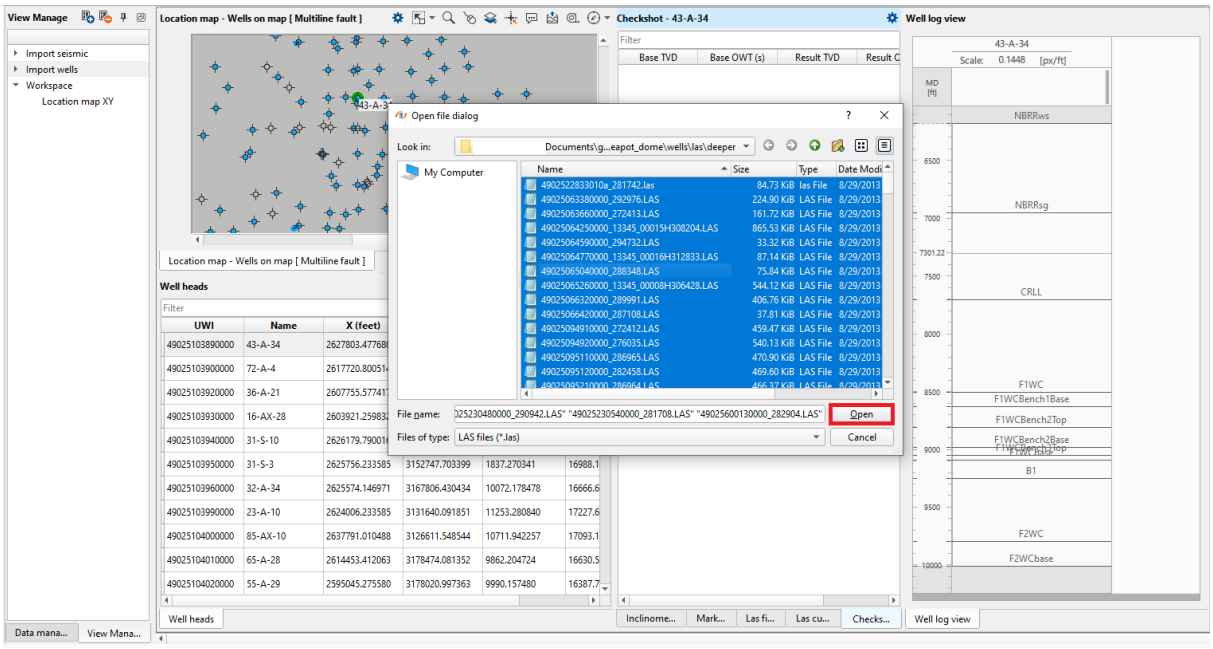


In the appeared dialog box locate the files and click open

The file dialog provides two additional options. Enable **Look for API within the filename** to extract the well API/UWI from the file name. The **Las collection** dropdown lets you add the curves to an existing collection or choose **Create new collection** to place them in a new one.



© 2026 Geomage 2003 LTD



Step 2: Create wells for unknown UWIs

If a LAS file refers to a UWI that is not in the project yet, g-Space asks whether to create these wells. Answer **Yes**, then confirm or edit the suggested identifier in the **New well name** dialog that opens for each file. The value you confirm becomes both the UWI and the name of the new well; leaving it empty skips that file.

For every well created this way, g-Space also fills in the well header from the LAS file, so the values do not have to be entered by hand afterwards. The following **Well heads** fields are filled in when the file provides them:

- **X** and **Y** - the surface location. Both coordinates must be present, otherwise the well is created without a location.
- **Kelly bushing** (KB).
- **Total depth, MD** - the total depth from the LAS header, or the deepest logged depth when the header does not state it.
- **Ground level** (GL). If the file has no ground level, the Kelly bushing value is used.
- **Completion date** - recognized when written as YYYY-MM-DD, YYYY/MM/DD or YYYYMMDD.

The values are taken from the well information (~W) and parameter (~P) sections of the LAS file, and values written in feet are converted automatically. If the logged data goes deeper than the total depth written in the LAS header, g-Space reports both values and asks *Should total depth be updated based on MD if it exceeds the LAS header value?* Choose **Yes** to store the deeper logged depth, or **No** to keep the header value.

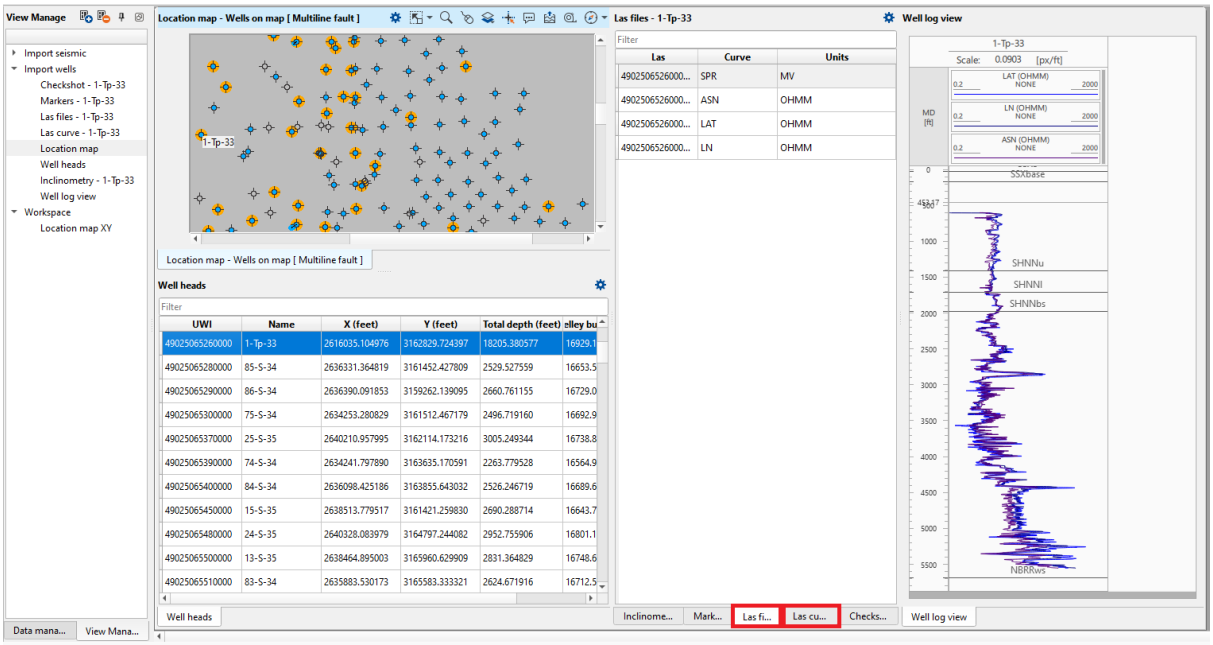
© 2026 Geomage 2003 LTD

Only wells created during this load receive header values. A well that already exists in the project keeps its current **X**, **Y**, **Kelly bushing** and depth values, and the LAS file simply adds its curves to that well.

To review the result, open the Wells data workspace 232 and check the **Well heads** table.

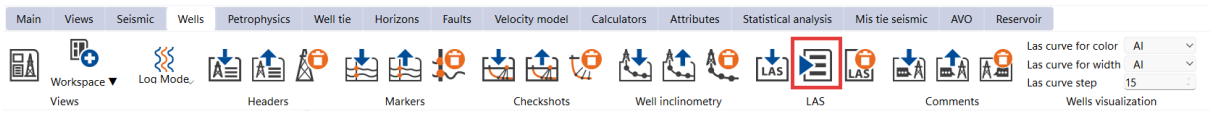
Step 3: Control loaded data

To check Well log data you have just loaded click on map view or in table below map. All loaded information will appear in table **LAS files** where you can find list of loaded curves. In the table **Las curve** you can check the log values. Also you can see the **log preview**



Log Loading Wizard (advanced)

The **Log Loading Wizard** gives you more control over the loading process.



In the **Log Loading Wizard** window, on the **Las files** tab, click the **+** button to add one or multiple LAS files. Every added file appears as a row with a **Las file** and a **Well name** column.

Each file has to be assigned to a well. g-Space fills in **Well name** automatically, matching the file against the project wells by UWI, then by API, then by the well name stored in the LAS header, and finally by a well name contained in the file name. If nothing matches, the cell stays

© 2026 Geomage 2003 LTD

Undefined and you have to select the well yourself in that cell, from a dropdown that lists **Undefined** and the wells already present in the project and that can be searched by typing. Wells cannot be created from this page, so a missing well has to exist in the project before it can be picked. If any file is still unassigned when you press **Finish**, g-Space reports *Some LAS files are not assigned a well*, lists the files concerned and keeps the wizard open.

The **Parameters** group under the table sets **Depth type**, **Depth units** and **Datum** with its value for all added files at once; these settings can still be adjusted file by file on the next page. Once the files are added, press **Next**.

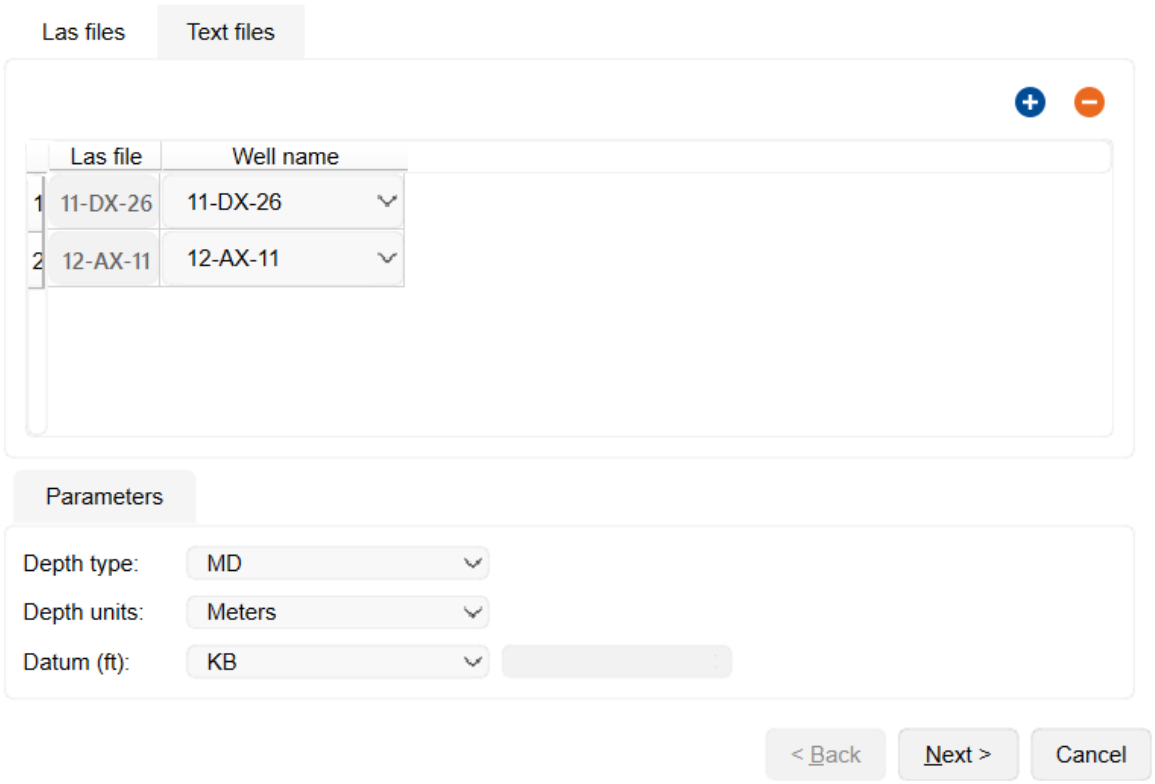
A separate **Text files** tab is available for importing log data from `.txt` / ASCII files. It uses the same curve-review workflow, with an added separator and column mapping so the wizard can read the curves from a plain text file.

When you add a `.txt` / ASCII file, an **Import log** column-mapping dialog opens. g-Space analyses the file and pre-fills the mapping automatically, so a standard log file can be imported with minimal manual editing:

- **Separator type** - Tab delimited, Space delimited or CSV, detected from the file content.
- **Headers row** and **Start row** - the header line and the first data row, detected from the file structure.
- The **MD** (depth) column and the log **Value** columns, bound to the matching file columns.
- **Mnemonics** read from the column headers, with **Units** filled in where available.

All detected values are only suggestions. You can change the **Separator type**, **Headers row** and **Start row**, edit the fields, mnemonics and units, and use the **Add** and **Remove** buttons to add or drop value fields. A preview of the parsed file lets you confirm the mapping before importing.

© 2026 Geomage 2003 LTD



The next page lists the curves of one file in the **Log curves** table. When several files were added, the **Log file** dropdown above the table switches between them, so the files are reviewed one at a time on this page and are then loaded together.

Each curve is listed with:

- **Load** - only the checked curves are imported. The checkbox in the column header selects or clears every curve of the current file at once; if nothing is checked, **Finish** reports *No curves selected*.
- **Curve name** (short code, e.g., GR, DT, NPHI).
- **Name from file** (original name as stored in the LAS).
- **Type** (curve type: Gamma Ray, Density, Resistivity, etc.).
- **Units**.
- **Comments**.
- **Start depth / Stop depth**.
- **Histogram preview** of curve distribution.

  Curve name, Type and Units can be changed manually, you can choose the options from the list or type it.

If the well already contains a curve with the same name, the wizard adds a number to the incoming curve, for example **DT 1**, so that the existing curve is kept. If you edit **Curve name**

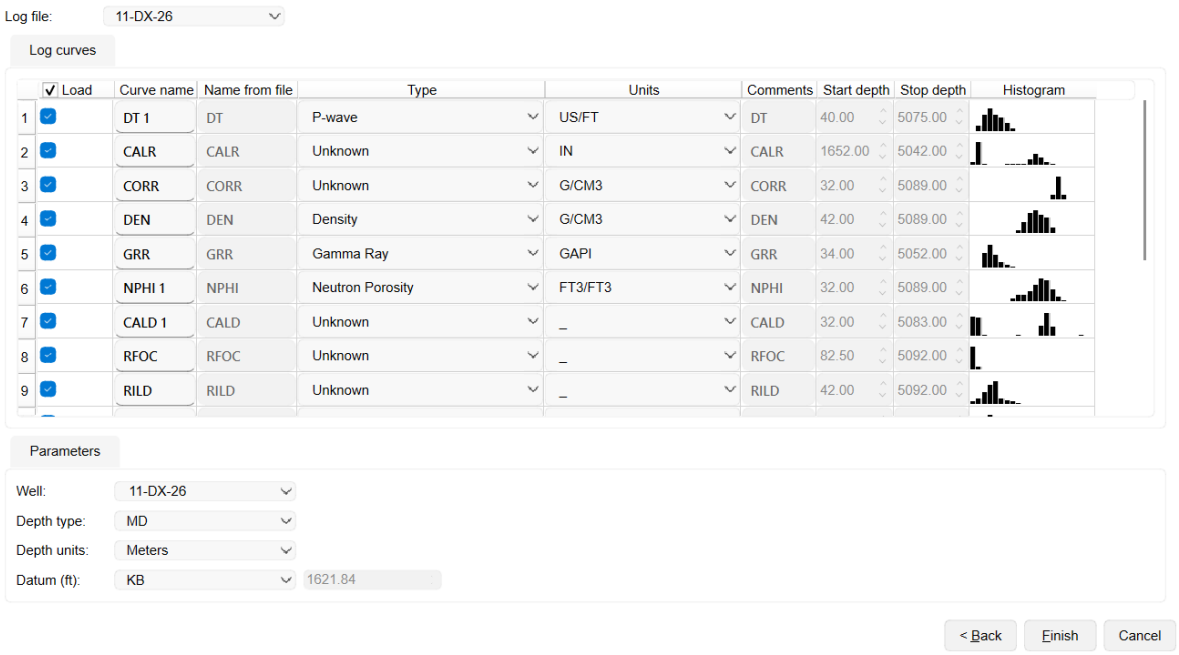
© 2026 Geomage 2003 LTD

back to a name that already exists in that well, the name is shown in red with the tooltip *Curve with the same name already exists*. Pressing **Finish** then asks *Some curves have the same names, do you want to rewrite them* and lists the well, the curve name and the number of curves concerned. Answer **Yes** to overwrite the existing curves, or **No**, which is the default, to return to the table and rename them.

When you assign or change a curve **Type** for a mnemonic (here or later in the Data Manager LAS curve table) and other wells contain the same mnemonic with a different type, g-Space asks *Apply this curve type to all curves with the same mnemonic in other wells?* Choose **Yes** to apply the selected curve type to every matching mnemonic in the other wells, or **No** to change only the current curve. This lets you classify a mnemonic once instead of repeating the assignment well by well.

At the bottom, you can set well-specific parameters that override the values read from the LAS header:

- **Well** (well name).
- **Depth type** (MD, TVD, etc.).
- **Depth units** (Meters, Feet).
- **Datum** — an editable combo box (KB, Sea level, Manual; or type your own) together with a numeric **Datum value**.



For more information about formats please refer to Well Data Formats 507

© 2026 Geomage 2003 LTD

5.2.7 Import checkshots

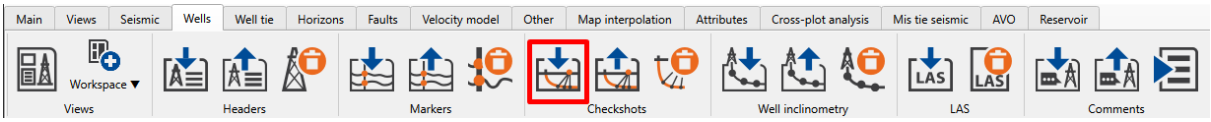
Note

Currently g-Space supports import checkshot data from KB datum.

Import checkshots steps:

Step 1: Locating the file

To load checkhots go to Ribbon 33 -> Wells bar 46 and click on **Load check shots** button

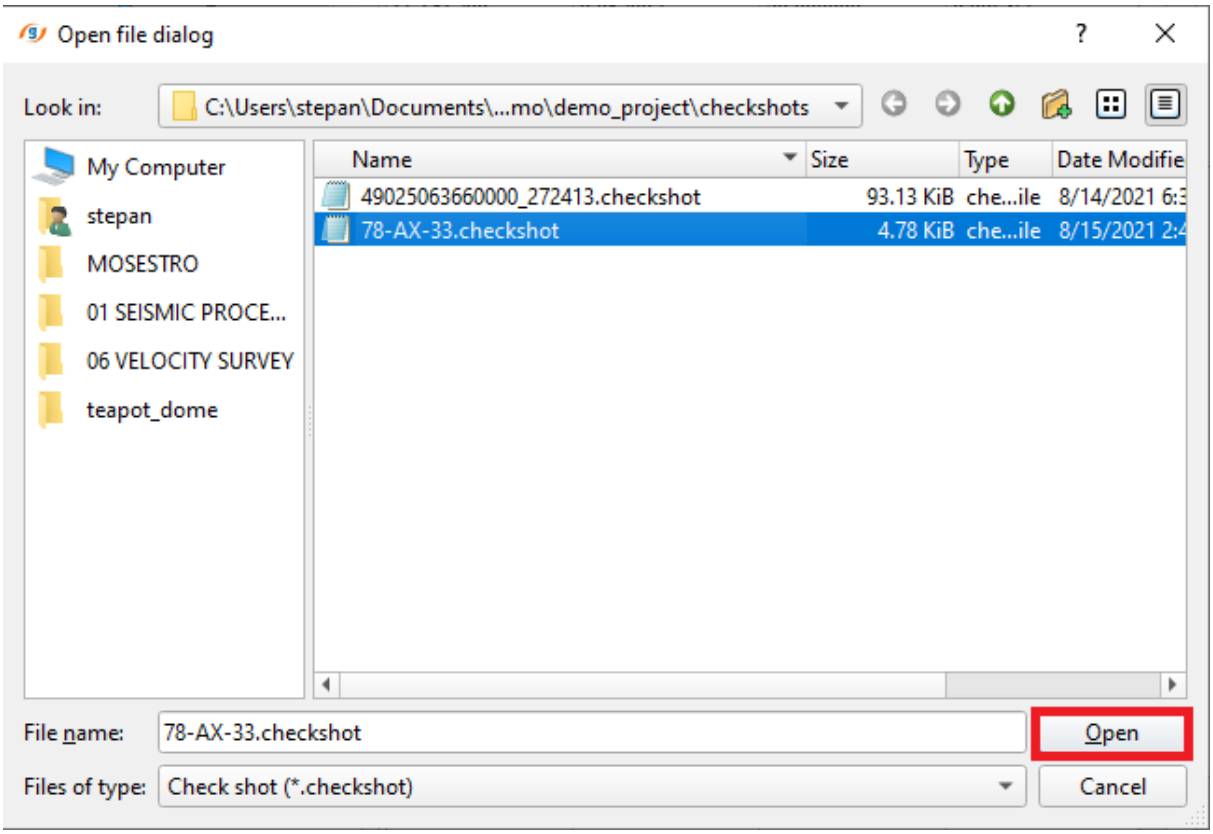


A standard checkshot file is a TAB-delimited ***.checkshot** file. It contains the following columns:

- **UWI** - well identifier (optional; if absent, the file name is used to match the well)
- **TVD** - true vertical depth (required)
- **Time** - travel time (required)

In the appeared dialog box locate the files and click open. In **Files of type**, keep **Check shot (*.checkshot)** for a standard checkshot table, or select **Petrel checkshots (*.txt *.dat *.las *.checkshot)** to list checkshots exported from Petrel:

© 2026 Geomage 2003 LTD



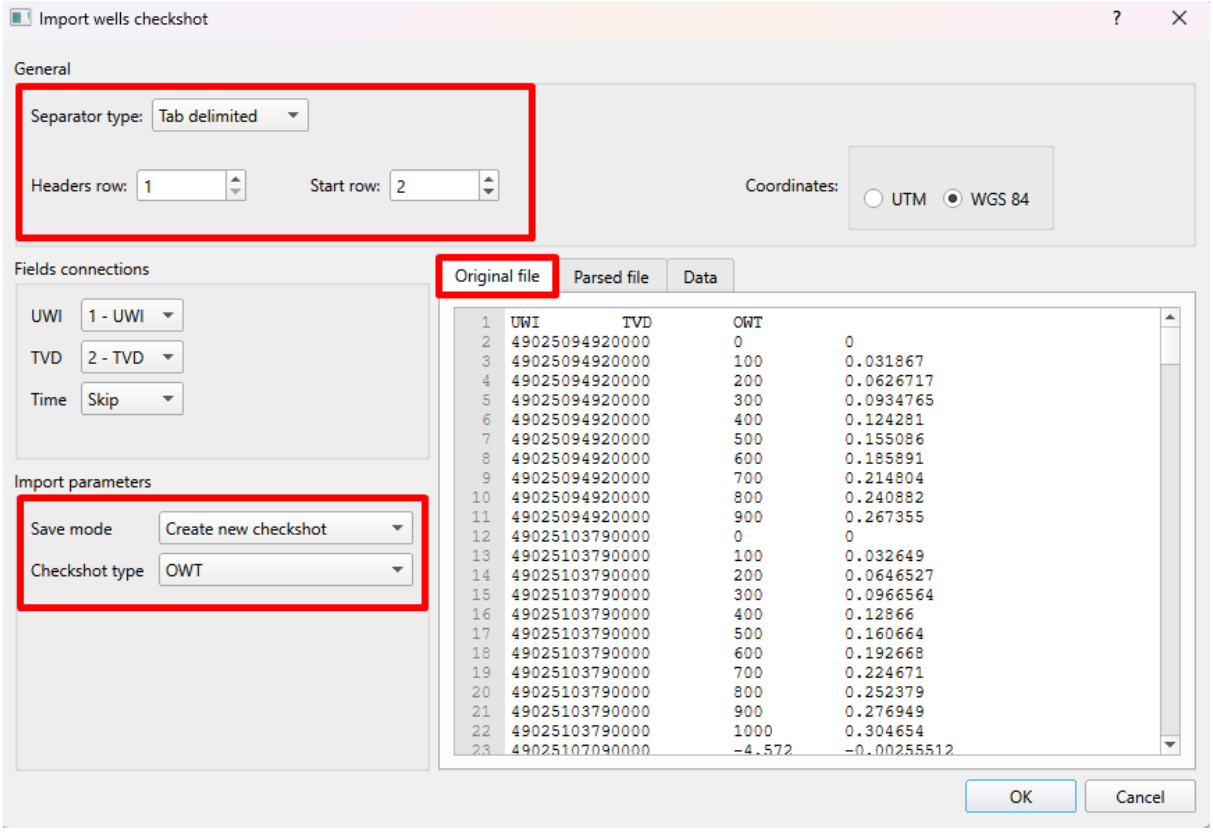
Step 2: Setting up reading parameters

g-Space will open file preview dialog were you can set the loading parameters such as: Separator type, Headers row, Start row, End row. Files in Petrel format are read directly and do not open this dialog.

In the **Original file** tab you will see how files looks like and adjust loading parameters accordingly. Also set the import parameters:

- **Checkshot type** - **OWT** (one-way time) or **TWT** (two-way time)
- **Time measure unit** - **s** or **ms**
- **Save mode** - **Create new checkshot** or **Append to current checkshot**

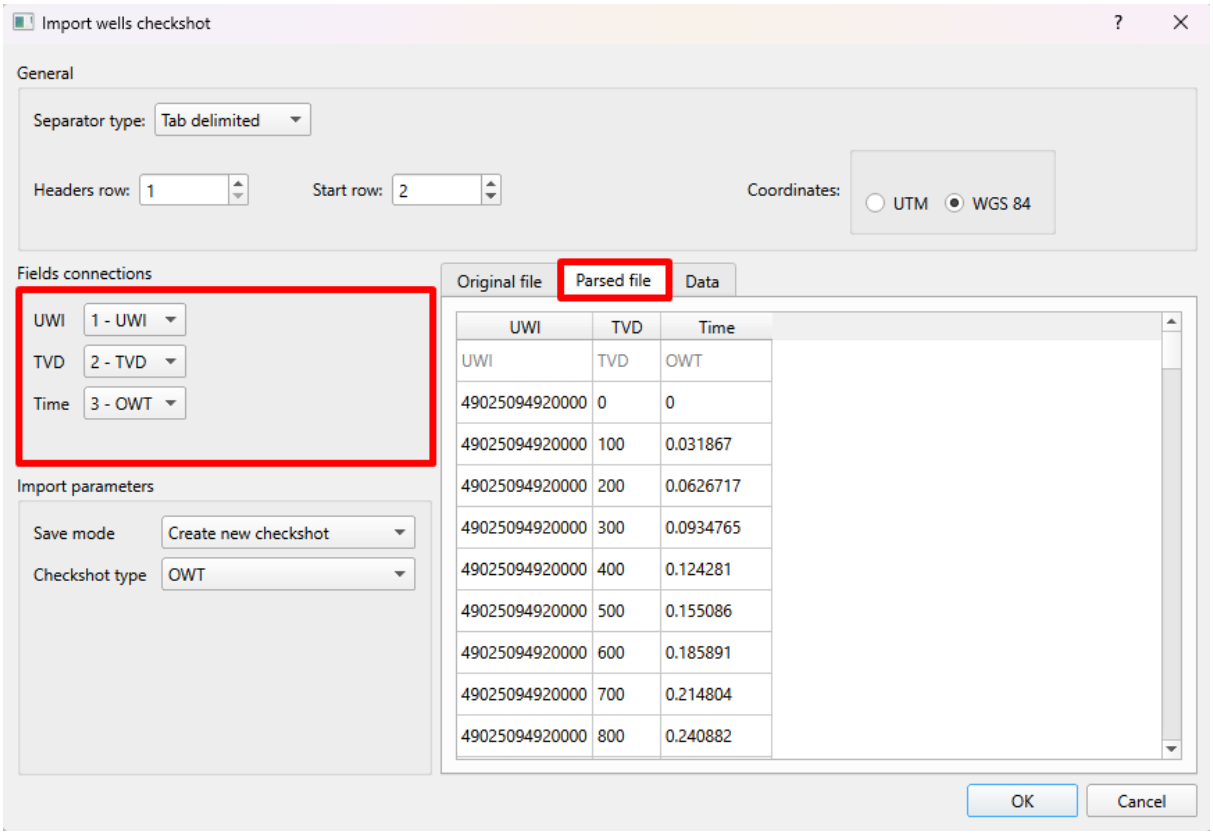
© 2026 Geomage 2003 LTD



Step 3: Checking reading parameters

Click on the **Parsed file** tab to check if you set reading parameters correctly and also you can decide which columns you want to add or skip

© 2026 Geomage 2003 LTD

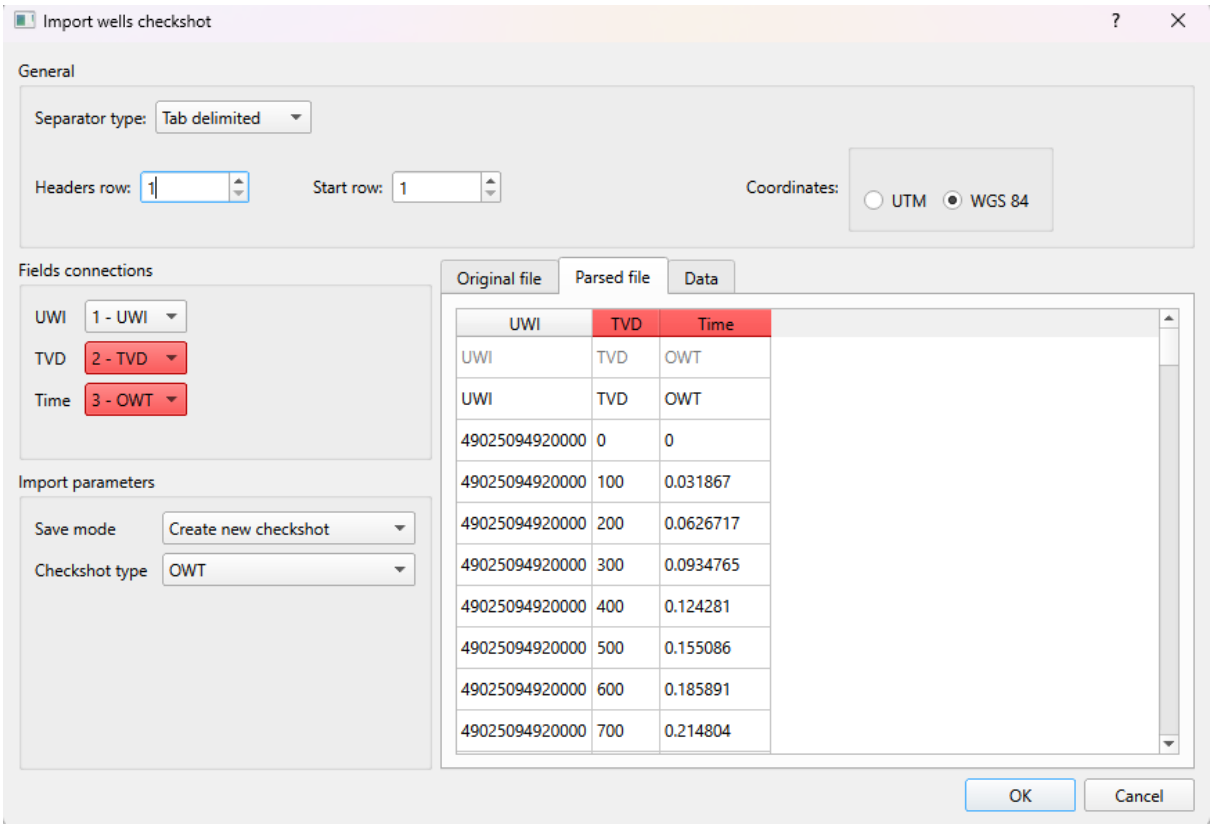


In the **Field connections** check the parameters. If you have files with additional information you can choose skip unnecessary columns from drop-down box.

Note

If some of the fields are colored RED, it means that data doesn't have required type or reading parameters were set incorrectly

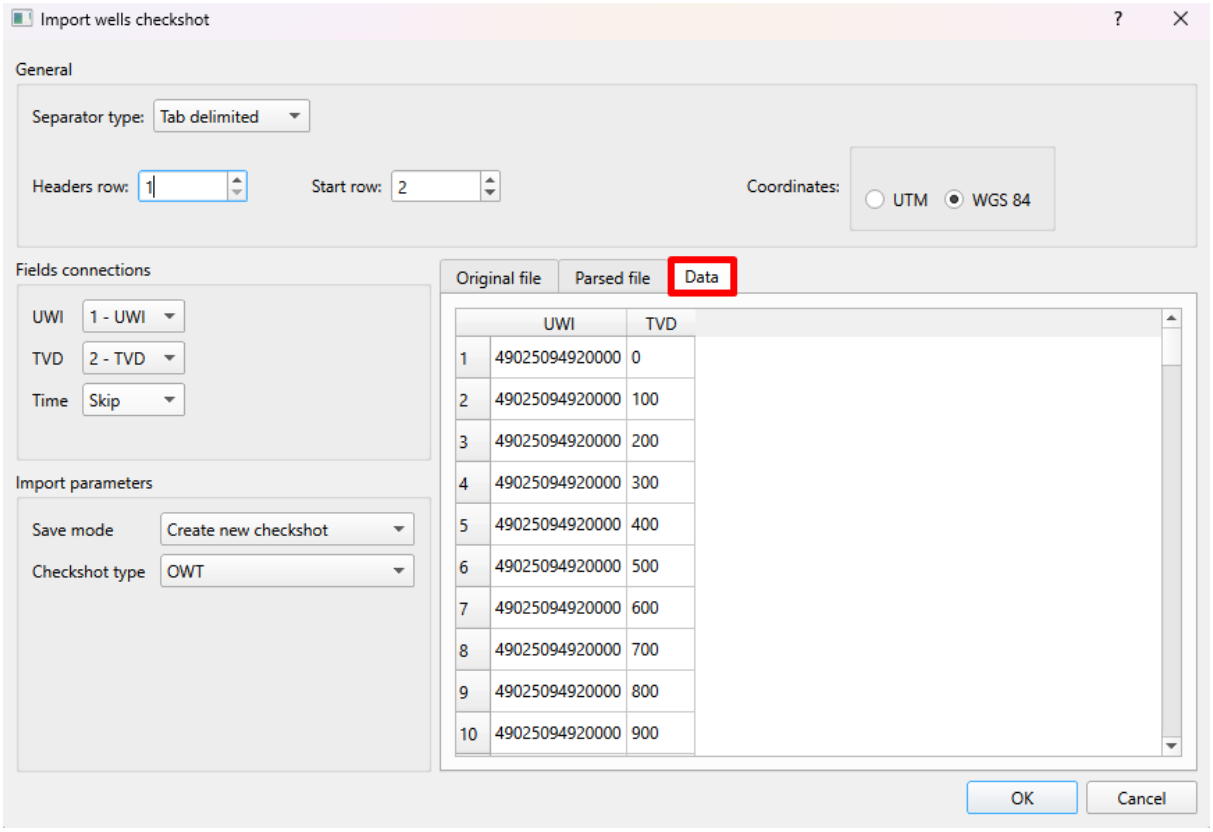
© 2026 Geomage 2003 LTD



Step 4. Checking the loading settings

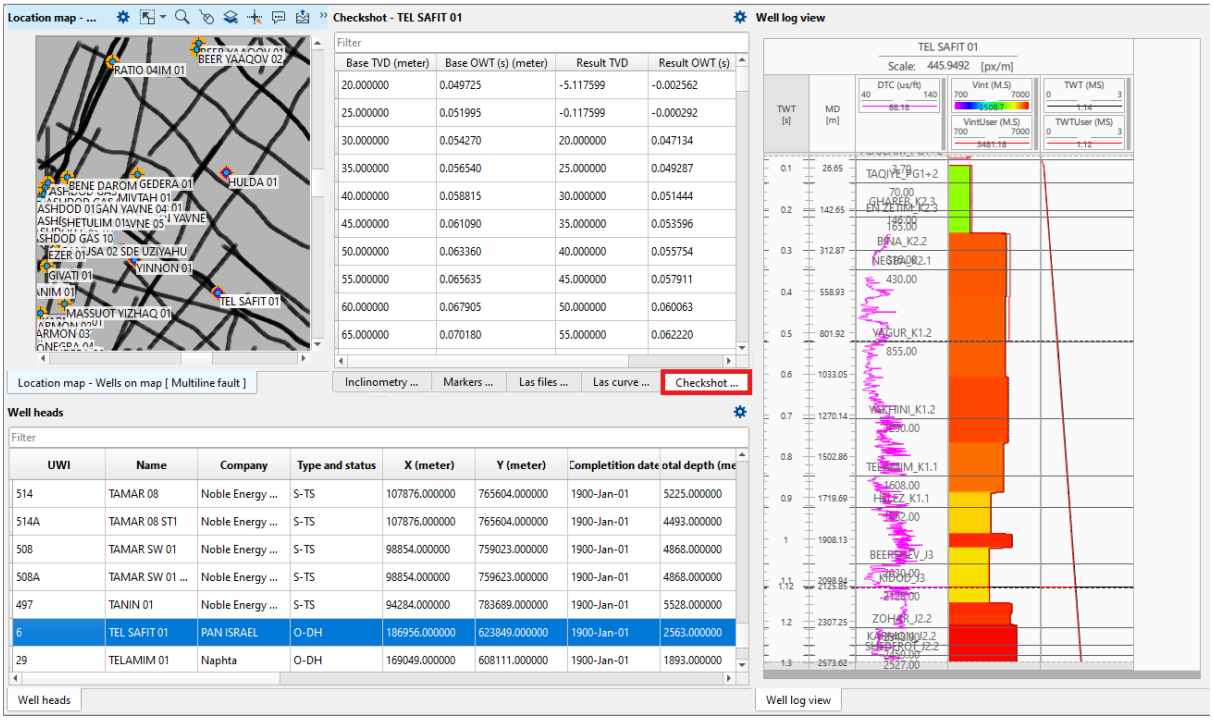
In Data tab you can see if all settings were done correctly and all necessary columns were included. After that click OK

© 2026 Geomage 2003 LTD



You can evaluate loaded data by clicking on map view or in table below map. All loaded information will appear in table **Checkshot** where you can check the values. Also you can see the data in **log preview** to the right

© 2026 Geomage 2003 LTD



Importing checkshots from Petrel format

Checkshots exported from Petrel are loaded with the same **Load check shots** button. In the file dialog set **Files of type** to **Petrel checkshots (*.txt *.dat *.las *.checkshot)** and select one or more files.

g-Space recognizes a Petrel checkshot file by its content: the first non-empty line must be a comment line that starts with **# Petrel checkshots format** (letter case does not matter). Such a file is read directly, so the reading parameters dialog of the steps above does not appear and the **Checkshot type**, **Time measure unit** and **Save mode** settings are not needed.

The file declares its columns inside a **BEGIN HEADER** ... **END HEADER** block, with one column name per line; the data starts on the line after **END HEADER**. Three columns are required:

- **Well** - well name or UWI, matched to the wells of the project
- **MD** - measured depth
- **TWT picked** - two-way time in milliseconds

A column name may also carry the Petrel type prefix, for example **FLOAT,MD**. g-Space converts the measured depth to TVD along the well trajectory and stores the time as one-way time in seconds. Each imported file creates a new checkshot named after the file; if a checkshot with that name already exists, a number in brackets is added to the new name.

© 2026 Geomage 2003 LTD

Petrel files and standard checkshot tables can be selected together: the Petrel files are read directly and the remaining files are loaded through the steps above.

Note

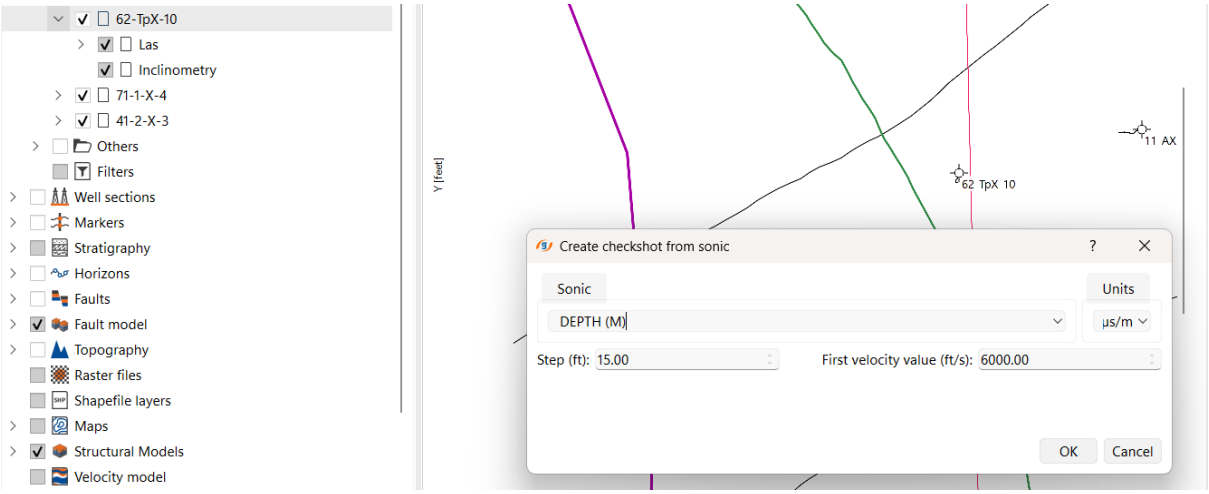
Rows of a well that is missing from the project are skipped, and g-Space reports the file and the unknown well name when the import finishes. A file whose header block is not closed, or which lacks one of the three required columns, is reported as containing errors.

Checkshots from Sonic log

If a well has a sonic log, a checkshot can be created from it instead of importing a file. Select the well, double-click it on the map and open the **Create checkshot from sonic** dialog, then set:

- **Sonic** - the sonic curve to use
- **Units** - the sonic units, **us/m** or **us/ft**
- **Step** - the depth step (in m or ft, according to the project units)
- **First velocity value** - the starting velocity above the first sonic sample

Click **OK** to create the checkshot from the sonic curve.



For more information about checkshot format please refer to Well Data Formats 507

5.2.8 Import comment log

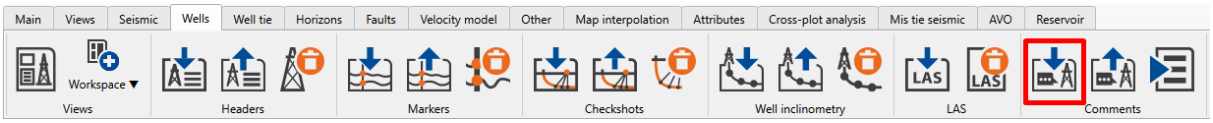
Comment logs are depth-referenced text annotations attached to a well - for example lithology descriptions, drilling notes, formation tops or fluid shows. They are matched to a well by its name/UWI and displayed alongside the well logs at their reference depth.

Import comments steps:

© 2026 Geomage 2003 LTD

Step 1: Locating the file

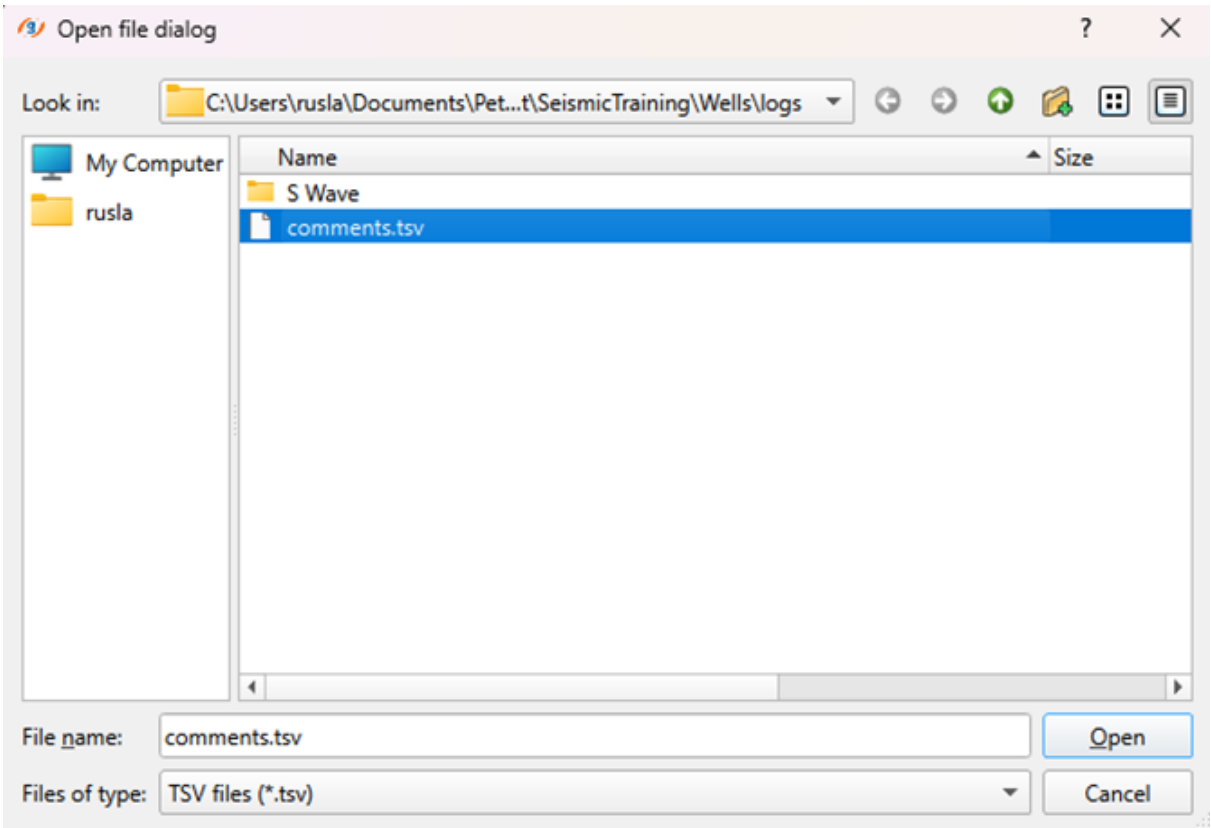
To load comments go to Ribbon 33 -> Wells bar 46 and click on **Import wells comments from text file** button



The comment file is a TAB-delimited ***.tsv** file with the following required columns:

- **UWI** - well identifier
- **MD** - measured depth of the comment
- **Comment** - the comment text

In appeared open file dialog locate comments file and click open

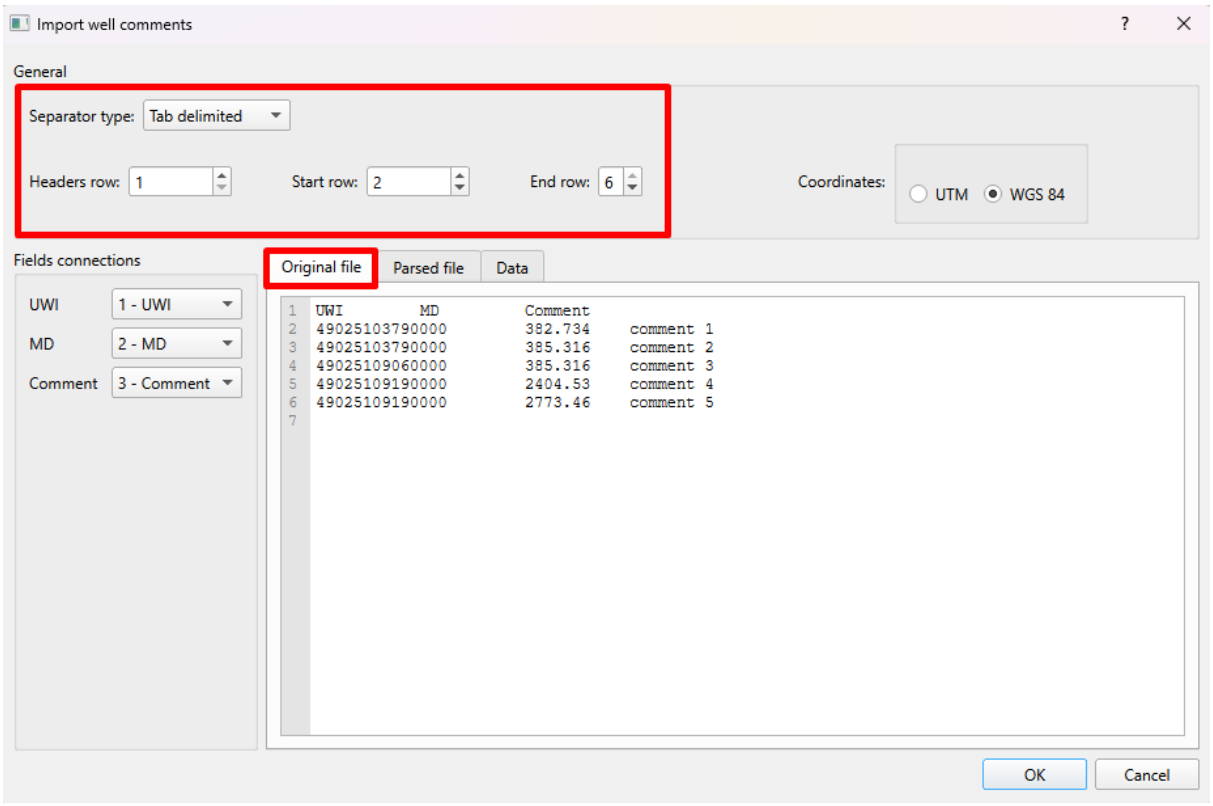


Step 2: Setting up reading parameters

g-Space will open file preview dialog were you can set the loading parameters such as: Headers row, Start row, End row. The separator is fixed to TAB for this importer.

© 2026 Geomage 2003 LTD

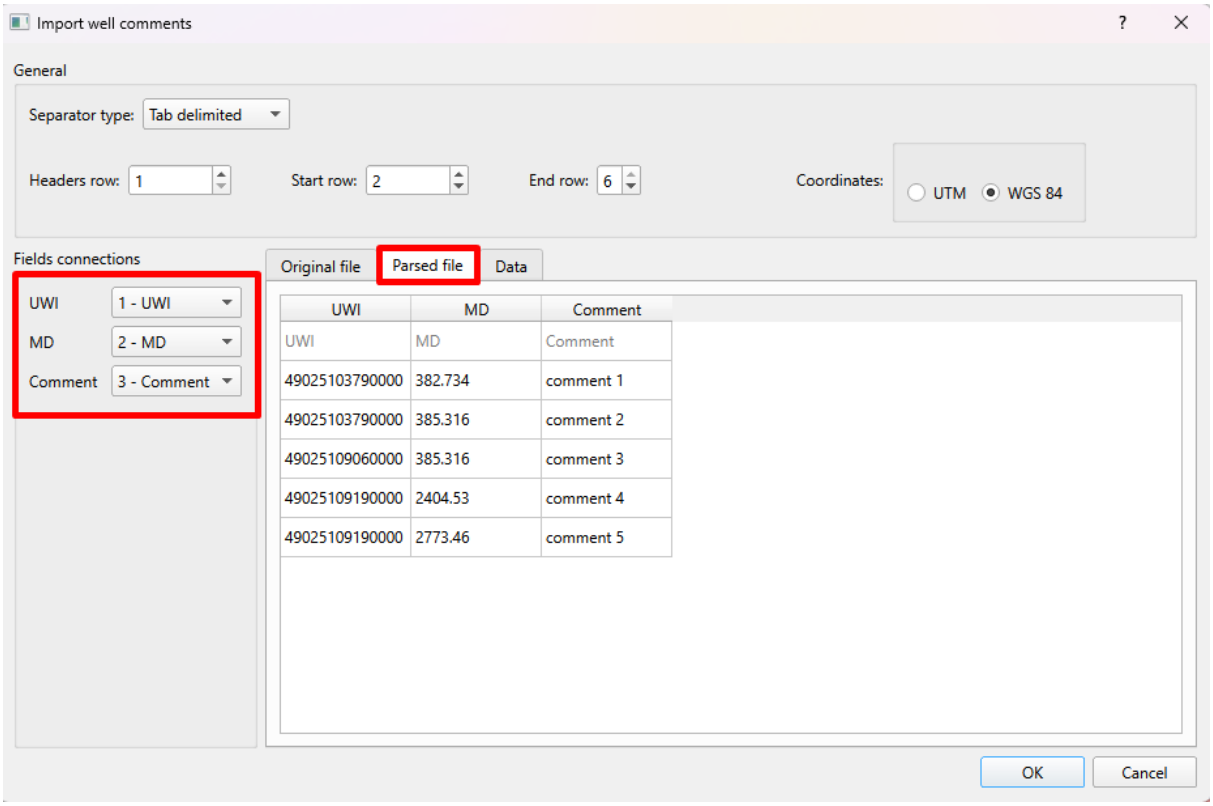
In the **Original file** tab you will see how files looks like and adjust loading parameters accordingly



Step 3: Checking reading parameters

Click on **Parsed file tab** to check if you set reading parameters correctly and also you can decide which columns you want to add or skip

© 2026 Geomage 2003 LTD

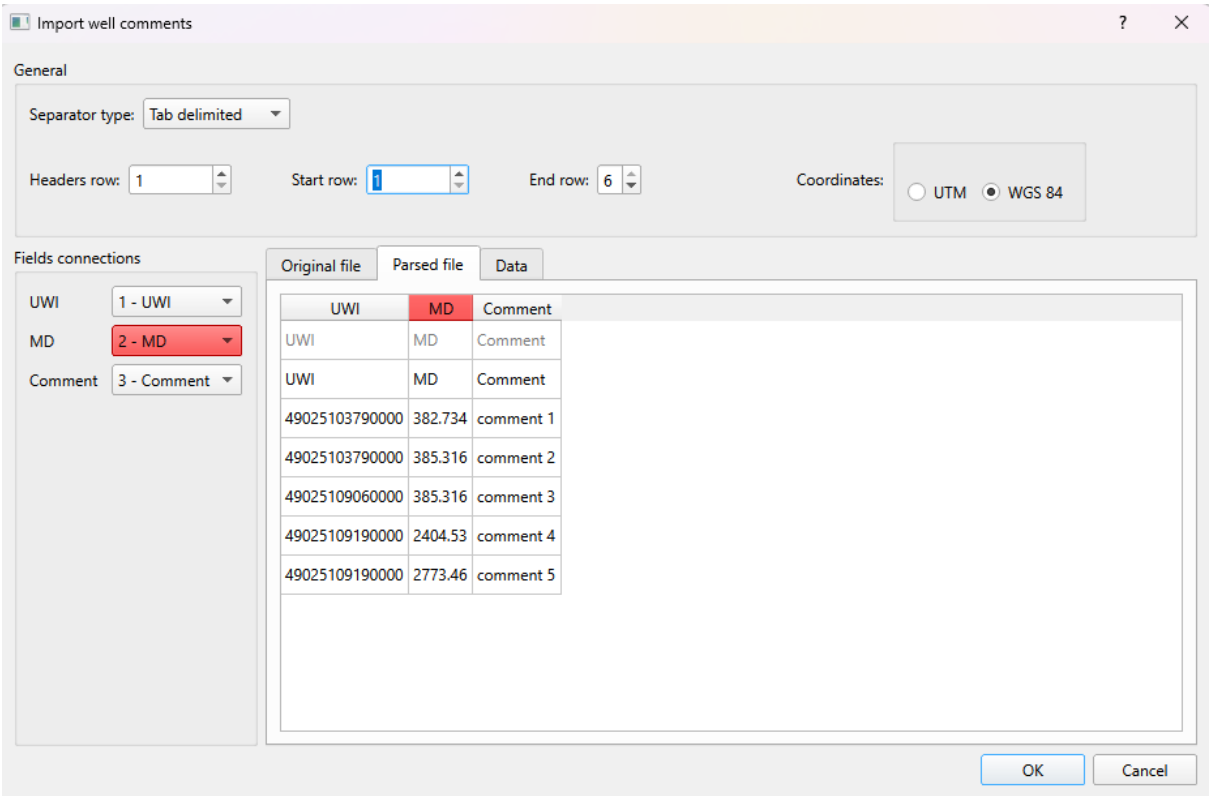


In the **Field connections** check the parameters. This importer expects exactly the **UWI**, **MD** and **Comment** columns; additional columns are not allowed.

Note

If some of the fields are colored RED, it means that data doesn't have required type or reading parameters were set incorrectly

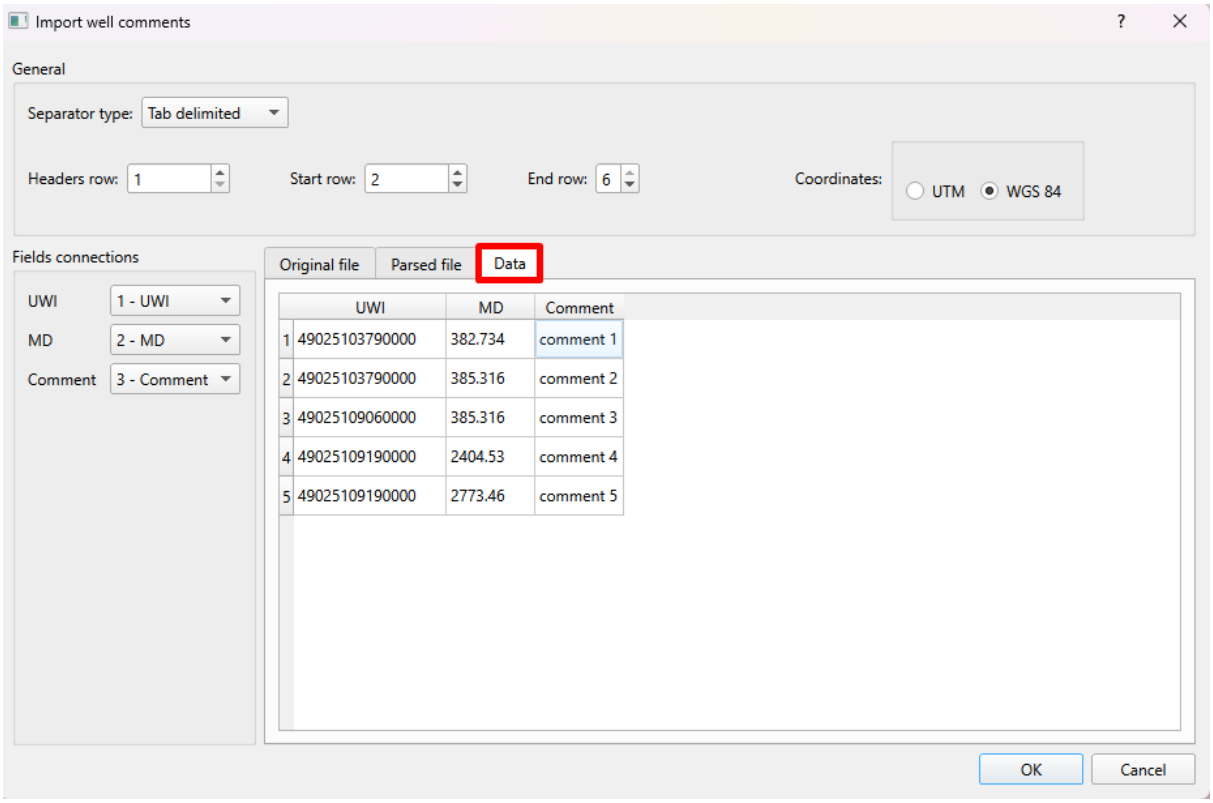
© 2026 Geomage 2003 LTD



Step 4. Checking the loading settings

In Data tab you can see if all settings were done correctly and all necessary columns were included. After that click OK

© 2026 Geomage 2003 LTD



After import the comments are attached to the matching well by its name/UWI and are shown alongside the well logs at their reference depth.

For more information about comment log format please refer to Well Data Formats 507

5.2.9 Import DLIS

DLIS (Digital Log Interchange Standard) is a binary industry format for well-log data. g-Space reads DLIS `.dlis` files through the same file dialog used to load LAS 407 files, so DLIS curves arrive as ordinary well log curves. This topic covers importing DLIS into a g-Space project.

Purpose

Load well-log curves stored in the binary DLIS format into the project's wells. Standard one-dimensional curves are loaded like LAS curves; multi-dimensional (array) channels, such as an NMR T2 distribution, are read natively so their per-depth arrays are preserved. Import only - g-Space does not write DLIS.

© 2026 Geomage 2003 LTD

Launching the import

Start the well-log import you normally use for LAS files 407. In the file dialog, set the file filter to DLIS (or **All Files**) and select one or more `.dlis` files. DLIS files are read through the same import as LAS, so no separate menu entry is needed.

Step-by-step

1. Open the LAS / well-log import and, in the file dialog, choose the `.dlis` file(s) to load.
2. g-Space reads the DLIS frames and channels. Standard 1D curves are extracted as normal well log curves.
3. Multi-dimensional array channels (for example, an NMR T2 distribution) are read natively and attached to the well, keeping their two-dimensional per-depth shape.
4. Confirm the load. The imported curves appear under the target well, alongside curves brought in from LAS 407 files.

Notes

- DLIS import is read-only. There is no export back to DLIS; export well curves as LAS instead.
- Multi-dimensional (array) channels keep their full per-depth arrays; ordinary single-value curves load exactly as they would from LAS.
- DLIS support may be unavailable in some builds or platforms. If DLIS cannot be loaded in your installation, convert the file to LAS with an external tool and import the LAS instead.

See Also

Import LAS files 407 | Well Data Formats 507

5.2.10 Import WITSML

WITSML (Wellsite Information Transfer Standard Markup Language) is an XML-based industry standard for exchanging well data. g-Space imports WITSML `.xml` files through a dedicated wizard. This topic covers importing WITSML into a g-Space project.

© 2026 Geomage 2003 LTD

Purpose

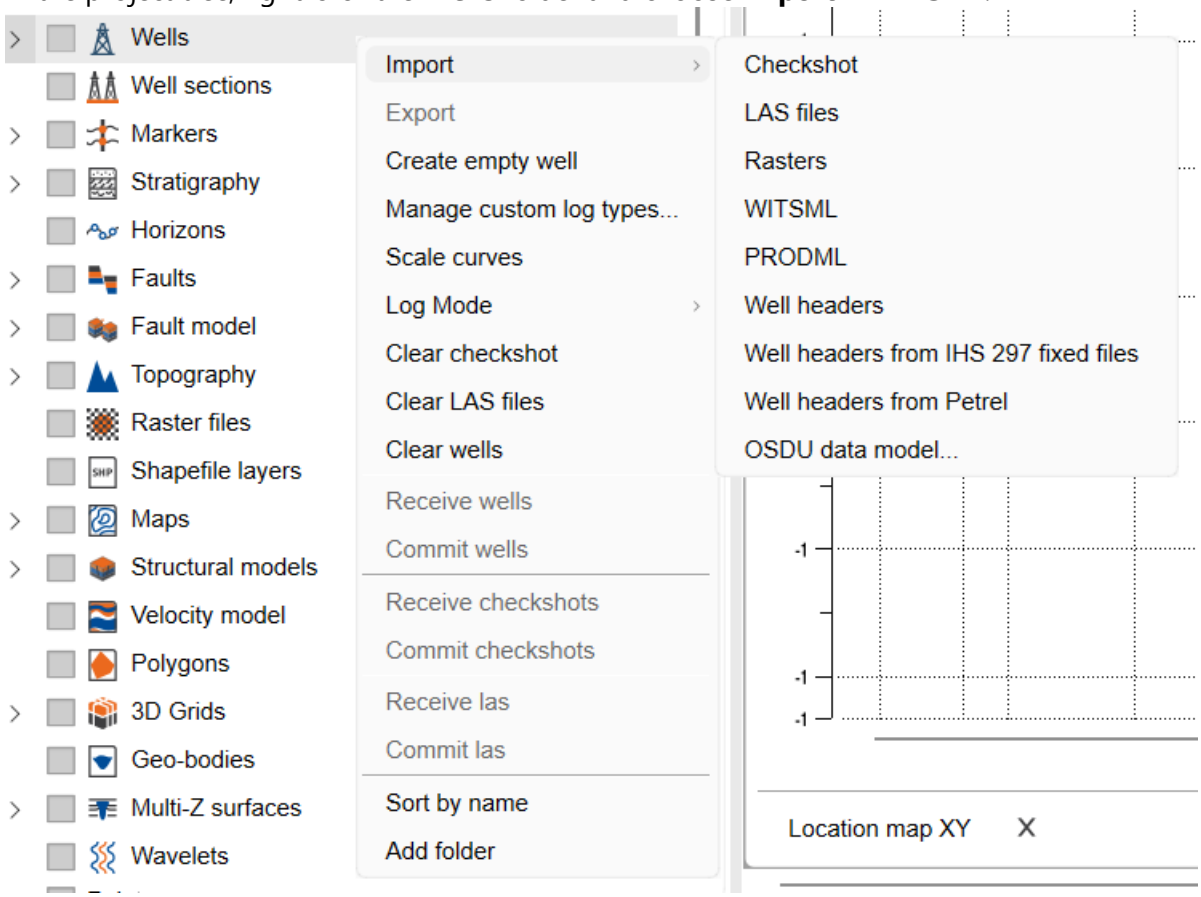
Load well data from WITSML documents into the project's Wells folder. A single WITSML file can carry several kinds of object, and the wizard imports them together:

- **Wells and wellbores** - the well hierarchy and its bores.
- **Well headers** - identifying and location information such as API number, country, operator, coordinate reference system, depth datum and surface location.
- **Logs and curves** - log curves with their mnemonics, units and sample values.
- **Deviation / trajectory** - trajectory stations with measured depth, TVD, inclination, azimuth and location.
- **Formation markers** - marker names with depth (MD / TVD) and geological attributes.

Import only - g-Space does not write WITSML. The supported WITSML version is 1.4.1.1.

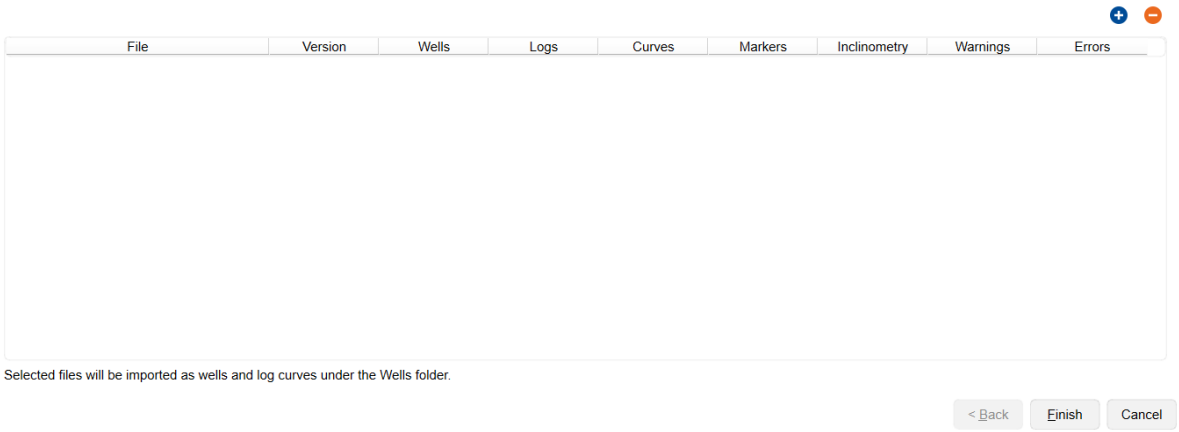
Launching the import

In the project tree, right-click the **Wells** folder and choose **Import** > **WITSML**.



The **Import WITSML files** wizard opens on a page with an empty file table. The plus and minus buttons above the table (**Add WITSML files** and **Remove selected files**) build the list of files to import.

© 2026 Geomage 2003 LTD



Step-by-step

1. Press the plus button and, in the **Open WITSML files** dialog, select one or more WITSML `.xml` files.
2. Each file is parsed and listed in the table. Review the per-file columns: **File**, **Version**, **Wells**, **Logs**, **Curves**, **Markers**, **Inclinometry**, **Warnings** and **Errors**. **Inclinometry** counts the trajectories found in the file.
3. Use the minus button to drop any file you selected by mistake. Selected files are imported as wells and log curves under the Wells folder.
4. Finish the wizard to import. If none of the selected files contain wells, logs, trajectories or markers, g-Space warns before continuing.
5. Review the **WITSML Import Report** that opens after import.

The import report

The **WITSML Import Report** summarises what was loaded. A header line shows the source, version and totals, and the tree below groups the results into **Imported wells**, **Imported curves**, **Imported trajectories**, **Imported formation markers**, **Warnings** and **Errors**. Use **Export to text...** to save the report to a file.
When the import finishes, g-Space also shows a short completion message with the totals for the run - the wells created and the curves, trajectory stations and markers added - followed by a list of anything that was skipped. The well total counts only newly created wells; when the import added data to wells that were already in the project, an extra line reports how many existing wells were updated.

Notes

© 2026 Geomage 2003 LTD

- The importer supports WITSML version 1.4.1.1. Files that declare a different version are reported as an error and are not imported.
- WITSML import is read-only; there is no export back to WITSML.
- g-Space has no separate wellbore object. When a WITSML well contains several wellbores, a separate well is created for each of them and named **[Well Name] - [Wellbore Name]**; the wellbore name and identifier are kept as metadata on that well.
- Imported objects are matched against the wells already in the project by the WITSML well identifier - combined with the wellbore identifier when the file names a wellbore - and then by the well name or the **[Well Name] - [Wellbore Name]** form, ignoring letter case. A file that carries only logs, trajectories or markers can therefore be imported after the wells themselves, and a file that repeats a well already in the project updates that well in place: its curves, trajectory stations and markers are added to the existing well instead of a second well being created. Objects whose well still cannot be found are skipped and listed in the message shown at the end of the import.
- Warnings (for example, missing CRS, missing depth datum, curves without units, or stations lacking usable data) do not stop the import - check the report to see which objects were skipped.

See Also

Import wells 391 | Well Data Formats 507 | Import PRODML 430

5.2.11 Import PRODML

PRODML is an Energistics data-exchange standard for production and well data. g-Space reads PRODML records from JSON `.json` files through a dedicated import wizard that previews the content and reports the outcome before anything is written to the project.

Purpose

PRODML import is a one-way import. It brings two kinds of data into the current project: well comments, which are attached to existing wells matched from the PRODML records, and point sets, which are built from records that carry coordinates but do not match any well. Wells are never created by this import.

Launching the import

Start the **Import PRODML files** wizard. On the first page, use **Add** to choose one or more `.json` files and **Remove** to drop a selected file. Each added file is parsed immediately and

© 2026 Geomage 2003 LTD

listed with its per-file counts: **Records**, **Matched wells**, **Comments planned**, **Points planned**, **Skipped** and **Errors**.

Step-by-step

1. Press **Add** and select the PRODML `.json` files to import. Repeat to build the list; each file is parsed as it is added.
2. Review the per-file counts in the table to confirm how many records were read, how many wells were matched, and whether any files reported errors.
3. Continue to open the preview. The preview lists every record with its source object type and id, the matched well (or a "no match" note), the record **Kind**, and the reason describing how it will be handled. A summary line totals files, records, matched wells, points and skipped records.
4. Confirm the preview to apply the import. Records that match an existing well become well comments (depth-bearing when a depth is present); records with coordinates but no matching well are routed to a PRODML import point set; the rest are skipped.
5. Read the import report. It groups the outcome into counters (parsed records, matched to wells, well comments created, depth-bearing well comments, point sets created, points created), issue buckets and records by kind. Use **Export to text** to save the report to a `.txt` file if needed.

Notes

- Wells are matched case-insensitively against existing wells using the PRODML name, UWI, API, reporting-entity name and any aliases. If no existing well matches, no well is created.
- A record that has no matching well but does carry coordinates is imported as a point instead of a comment. Point sets are created only when at least one such orphan point lands.
- A record with neither a matching well nor coordinates is skipped and listed in the report, together with any unsupported object types and parse errors.
- PRODML import is read-only; there is no PRODML export. Multiple files can be added at once and are aggregated into a single preview and report.

See Also

Import wells 391 | Import WITSML 427 | Well Data Formats 507

© 2026 Geomage 2003 LTD

5.2.12 Import from OSDU

OSDU is an open subsurface data platform whose records are exchanged as JSON `.json` documents. g-Space reads OSDU records with a dedicated import dialog that classifies the content by record kind, lets you choose what to bring in, and maps the selected records into the project's wells.

Purpose

OSDU import is a one-way import. A single import can carry several kinds of objects at once: wells and wellbores, deviation (trajectory) data, well logs and their curves, markers and tops, and point sets. The importer groups records by kind so you can review and select exactly what to load, then a field mapper matches or creates the corresponding g-Space wells and attaches the related data.

Launching the import

Start the OSDU import and point it at the source. The reader accepts a single OSDU record file, a manifest file, or a whole folder of `.json` files. The **Import OSDU data model** dialog opens, showing the source path and the number of OSDU records detected.

Step-by-step

1. Choose the source: a single OSDU JSON file, a manifest, or a folder of `.json` files. The dialog parses it and reports how many records were found.
2. Review the record groups. Records are laid out in collapsible groups by kind, such as **Wells**, **Wellbores**, **Trajectories**, **Well logs**, **Markers** and **Spatial records**. Each group lists its records with Name, Record id, Kind and Source file.
3. Select what to import. Use the per-record checkboxes, toggle a whole group with its header checkbox, or use **Select all** / **Select none**. The report area at the bottom summarises the detected records and any reader issues.
4. Press **OK** to run the import. The field mapper matches each selected record to an existing well by OSDU record id, by name, or by surface coordinates, and creates a new well when nothing matches.
5. Let the mapper attach the related data: wellbores are kept as metadata (extra wellbores under one well become sidetracks), trajectory stations populate the well path and

© 2026 Geomage 2003 LTD

inclinometry, well-log curves are added as a log file on the parent well, markers go to the marker set, and spatial records become point sets.

Notes

- Records whose kind is not recognised are grouped under Unsupported and are not mapped; nothing is dropped silently, so you can see them in the dialog and the report.
- Well logs that only reference an external LAS or DLIS file, and records without a resolvable parent well, are skipped and listed in the import report rather than mapped.
- Matching by surface coordinates uses a small tolerance, so a well already in the project may be reused instead of duplicated. The report lists how many wells were created, matched or added as sidetracks.
- OSDU import is read-only; there is no OSDU export from g-Space.

See Also

Import wells 391 | Well Data Formats 507

5.2.13 Import stratigraphy

A **stratigraphy** (stratigraphic column) describes the geological zones of a project, each with a name, an age and a display colour. g-Space reads a stratigraphic column from an `.xml` file. This topic covers importing that column into a project. Import is one-directional: stratigraphies are loaded from XML, there is no matching export.

Purpose

Load a stratigraphic column into the project so that geological zones and their ages are available for age-based colouring of markers, logs and cross-sections. The imported column is added to the project stratigraphy set and can be made the default. A stratigraphy may hold several nested columns that represent successive levels of the hierarchy (era, period, stage, and so on).

Launching the import

© 2026 Geomage 2003 LTD

In the Data manager 124, right-click the **Stratigraphy** node and choose **Load from XML**. The same menu also offers **Load default**, which loads the built-in International Stratigraphic Chart, and **Clear**.

Step-by-step

1. Right-click the **Stratigraphy** node and select **Load from XML**.
2. In the file dialog, set the filter to **XML files (*.xml)** and choose the stratigraphic column file.
3. The column is read: each zone contributes its name, age and colour, and any nested sub-columns are preserved as hierarchy levels.
4. When prompted, confirm or edit the **Stratigraphy name**. The name must be unique within the project; the file name is offered as the default.
5. The stratigraphy appears under the Stratigraphy node and is set as the current (default) column, refreshing age-based colouring across the project.

Notes

- The XML file must contain a `Content` / `Elements` tree of columns; each column carries a `Header` name and a list of zones, and each zone provides a `Name`, an `Age` and an optional `Color` (R, G, B).
- A file with no readable columns is rejected, and a name that already exists must be changed before the import completes.
- Stratigraphy handling is part of the geological interpretation license; the action reports that it is unavailable when that license is not present.

See Also

Import wells 391 | Well Data Formats 507

5.2.14 Import marker ages

Marker ages assign a geological age (and an optional type) to the markers of a marker set. g-Space reads them from a plain, tab-separated text `.txt` file with three columns: Marker, Age and Type. This topic covers importing that file; a matching export is also available (see Export marker ages 497).

© 2026 Geomage 2003 LTD

Purpose

Populate the ages of existing markers in a marker set from an external table. Only markers whose names already appear in the set are updated; the import matches on marker name and does not create new markers. Ages drive age-based colouring against the project stratigraphy.

Launching the import

In the Data manager 124, right-click the target **marker set** and choose **Import markers ages**.

Step-by-step

1. Right-click the marker set to update and select **Import markers ages**.
2. In the file dialog, set the filter to **Text file (*.txt)** and choose the ages file.
3. Each line is split on tabs and matched to a marker by the name in the first column. Matching markers receive the age from the second column and, when present, the type from the third.
4. Lines whose marker name is not found in the set, and lines missing a name or age, are skipped. The marker set is flagged as modified and the Data manager refreshes.

File format

The file is tab-separated with one marker per line. Columns are:

- **Marker** - the marker name, which must match a marker already in the set.
- **Age** - the numeric age value.
- **Type** - optional integer: `0` = Marker, `1` = Fault, `2` = Intrusion. When omitted, the type defaults to Marker.

This is the same layout produced by Export marker ages 497, so an exported file can be edited and re-imported. Marker ages are part of the geological interpretation license.

See Also

Export marker ages 497 | Import markers 403 | Well Data Formats 507

© 2026 Geomage 2003 LTD

5.2.15 Import well log images

Well log images are scanned or digitised raster pictures of paper well logs. g-Space imports these rasters and attaches them to a well over a measured-depth interval so that the image can be displayed alongside digital curves. Supported image formats are `.png`, `.jpg`, `.jpeg`, `.bmp` and `.xpm`. This topic covers importing them and adjusting the depth range of a raster that is already loaded; there is no export back to raster.

Purpose

Load a scanned well-log raster and pin it to a well between an MD begin and MD end value, so the image lines up with the well track. A single image can be loaded interactively, or many images can be loaded in bulk from XML manifest files that carry the well identifier, image file name and depth calibration.

Launching the import

Open the Data manager and use the well context menu. There are two entry points:

- Right-click a single well and choose **Load raster** to import one image with the **Open raster** wizard.
- Right-click the **Wells** root node and choose **Import** > **Rasters** to import many images at once from XML manifests with the **Open rasters** wizard.

Step-by-step: single image

1. In the **Open raster** wizard, press **Browse** and select an image file (filter `Images (*.png *.xpm *.jpg *.jpeg *.bmp)`).
2. In the **Preview** panel, correct the orientation if needed with the rotate buttons (↺ **90°**, **180°**, ↻ **90°**) or **Reset** to return to the original orientation.
3. Zoom with the mouse wheel and pan with the middle mouse button. Drag the edges or corners of the crop rectangle to trim the image to the useful log strip.
4. In the **Position** group, set **MD begin** and **MD end** (in the project distance units, m or ft) to the measured-depth range the image spans.
5. Press **OK** to add the raster to the well. If no valid image is loaded the wizard warns *Raster is not valid!*.

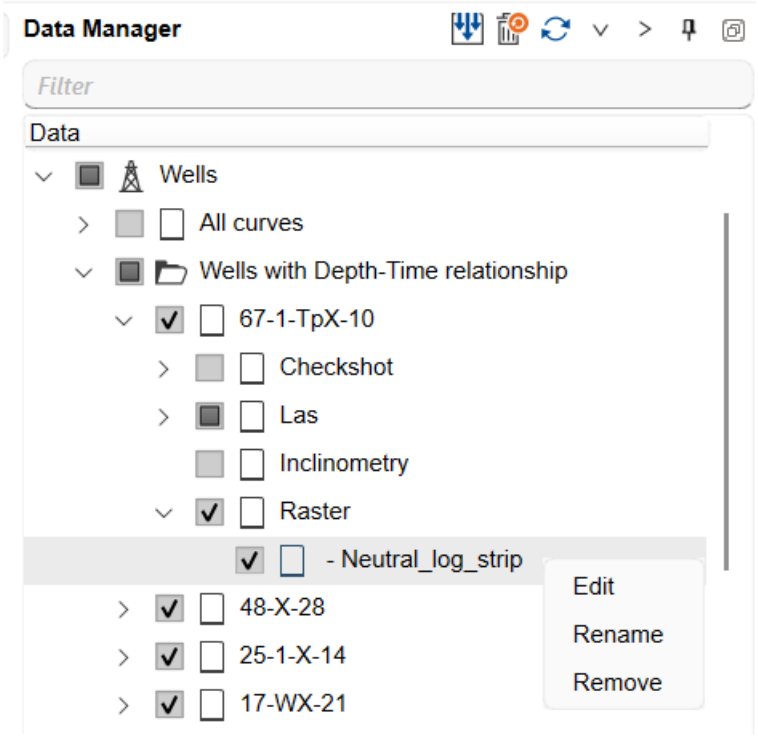
© 2026 Geomage 2003 LTD

Step-by-step: bulk import from XML

1. In the **Open rasters** wizard, press **Browse XML files** and select one or more `.xml` manifest files (filter `XML files (*.xml)`).
2. Each valid manifest is added to the table, showing the **XML file**, **API**, **Min MD** and **Max MD** read from the file. Remove a row with its minus button.
3. Each manifest names the image file to load (resolved next to the XML), the well identifier (API / UWI) used to match the well, and depth calibration points. Depths given in feet are converted to metres automatically.
4. Press **OK** to import all listed rasters. Wells whose identifier is not found in the project are skipped and reported; if no files were added the wizard warns *No files were selected*.

Change the depth range of a loaded raster

The measured-depth interval of a raster that is already attached to a well can be corrected without removing the image and importing it again. In the Data manager, expand the well down to its **Raster** group, right-click the raster and choose **Edit**. The same menu also holds **Rename** and **Remove** for that raster.



The **Edit raster** dialog opens. Its title also names the raster, and the **Position** group shows the values the raster currently uses:

© 2026 Geomage 2003 LTD

1. Change **MD begin** and **MD end**. Both labels carry the project distance units, **m** in a metric project and **ft** in an imperial one.
2. Press **OK** to apply the new interval, or **Cancel** to leave the raster as it was.

MD begin must be smaller than **MD end**; otherwise the dialog reports *MD begin must be less than MD end.* and stays open so the values can be corrected. Only the depth interval changes: the stored image and its cropping are kept, and the raster is repositioned and rescaled to the new interval wherever it is shown beside the well curves.

Notes

- Supported image formats are PNG, JPG, JPEG, BMP and XPM. The cropped raster is stored inside the project as PNG.
- In the single-image wizard the depth interval is entered manually; in the XML bulk wizard it is taken from the manifest calibration and the raster is cropped to that calibrated range.
- Import is one-way: rasters cannot be exported back to image files.

See Also

Import wells 391 | Well Data Formats 507

5.3 Import Horizons

g-Space supports several options importing horizons:

Navigate to the Interpretation 96 bar and use the **Horizons Management** group.

Option 1: Press load horizons binary button, locate file and press OK. g-Space will load binary horizons

IMPORTANT NOTE: Binary format contains a full copy of horizons exported from g-Space and it will overwrite all existing horizons

© 2026 Geomage 2003 LTD

Option 2: Press Import horizon ASCII button, locate one or more files and press OK.

Option 3: Press **Import horizon (charisma format)**, select one or more files and press OK. Select the **Is depth horizon** check box in the file dialog to import the files as depth horizons; leave it cleared to import them as time horizons.

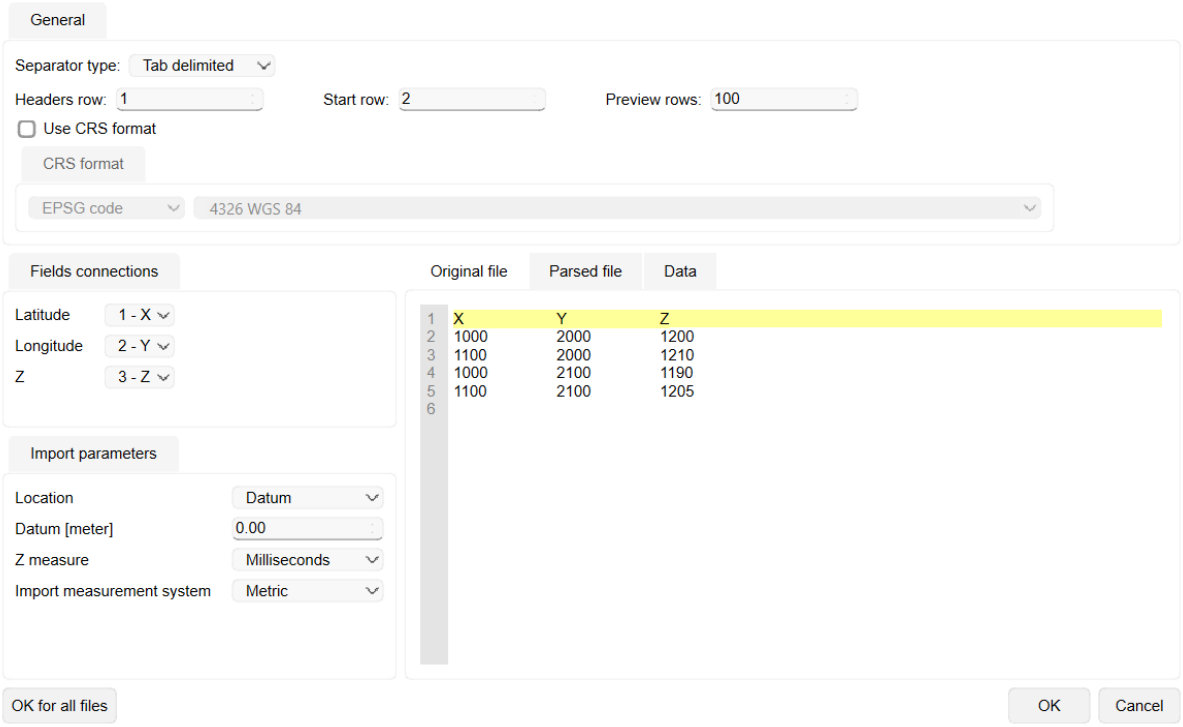
For more information about horizon format please refer to chapter Horizon Data Format 513

Option 4: Import horizons from the Data manager 124
In the Data manager 124, expand **Horizons** and right-click a **Time** or **Depth** folder. Choose **Import horizon** and then one of:

- **From ASCII...** - opens the Import horizon ASCII wizard.
- **From Charisma...** - imports horizons stored in Charisma format.

You can select several files at once; each file is imported into the selected domain folder, so g-Space does not ask you for the domain. When you import from Charisma, g-Space asks for the input **Datum** and, in the file dialog, for the source coordinate reference system and measurement system.
When you import from ASCII, g-Space opens the **Import horizon** wizard for the selected file. In it you connect the file columns in the **Fields connections** group and set **Location**, **Datum**, **Z measure**, **Import measurement system** and the source coordinate reference system of the file.
If you selected several files, the wizard opens again for the next file. Select **OK for all files** to apply the settings shown in the wizard to all the remaining files and import them without opening the wizard again, or **OK** to import only the current file and continue with the next one.

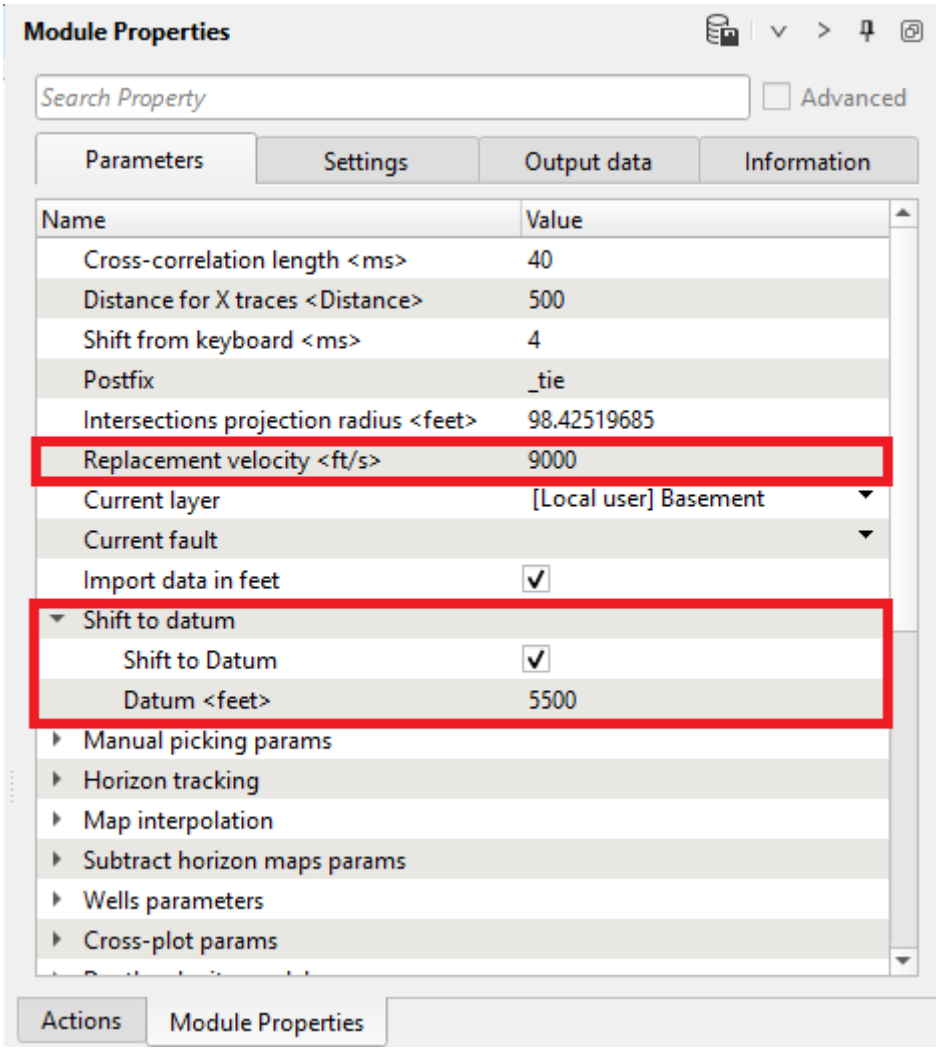
© 2026 Geomage 2003 LTD



Whichever option you use, when you import from Charisma or import several ASCII files at once, g-Space names the new horizons for you: from the name stored in the Charisma file, or from the file name for ASCII files. If a horizon with the same name already exists, g-Space keeps the existing horizon and adds a numeric suffix to the imported one instead of asking you to type a new name.
Importing horizons from ASCII adds them to the project and keeps the horizons you already have. Only **Load horizons binary** (Option 1) replaces the whole set of horizons, and it asks you to confirm before the previous ones are lost.

Before importing horizons make sure that the project **datum** and **replacement velocity** are the same for importing horizons otherwise in must be changed. To do this go to Module Properties 197 panel --> Parameters tab

© 2026 Geomage 2003 LTD



5.4 Import Faults

g-Space supports several options importing faults:

On the **Interpretation** tab, use the **Faults Management** group.

Option 1: Select **Load faults binary**, choose a fault file, and confirm. g-Space adds the imported faults to the existing faults. If an imported fault name is already in use, g-Space appends _2, _3, and so on so that both faults are kept. If the file cannot be parsed, g-Space reports that it could not load the fault file and leaves the existing faults unchanged.

© 2026 Geomage 2003 LTD

Option 2: Select **Import fault (charisma format)**, choose one or more files, and confirm.

Adjust the import settings as needed:

- **Imported EPSG code**: Displays the coordinate system of the imported data.
- **Import measurement system**: Allows selecting between Metric and other units.
- **Use converting on import**: Enables automatic conversion of coordinate or measurement systems.

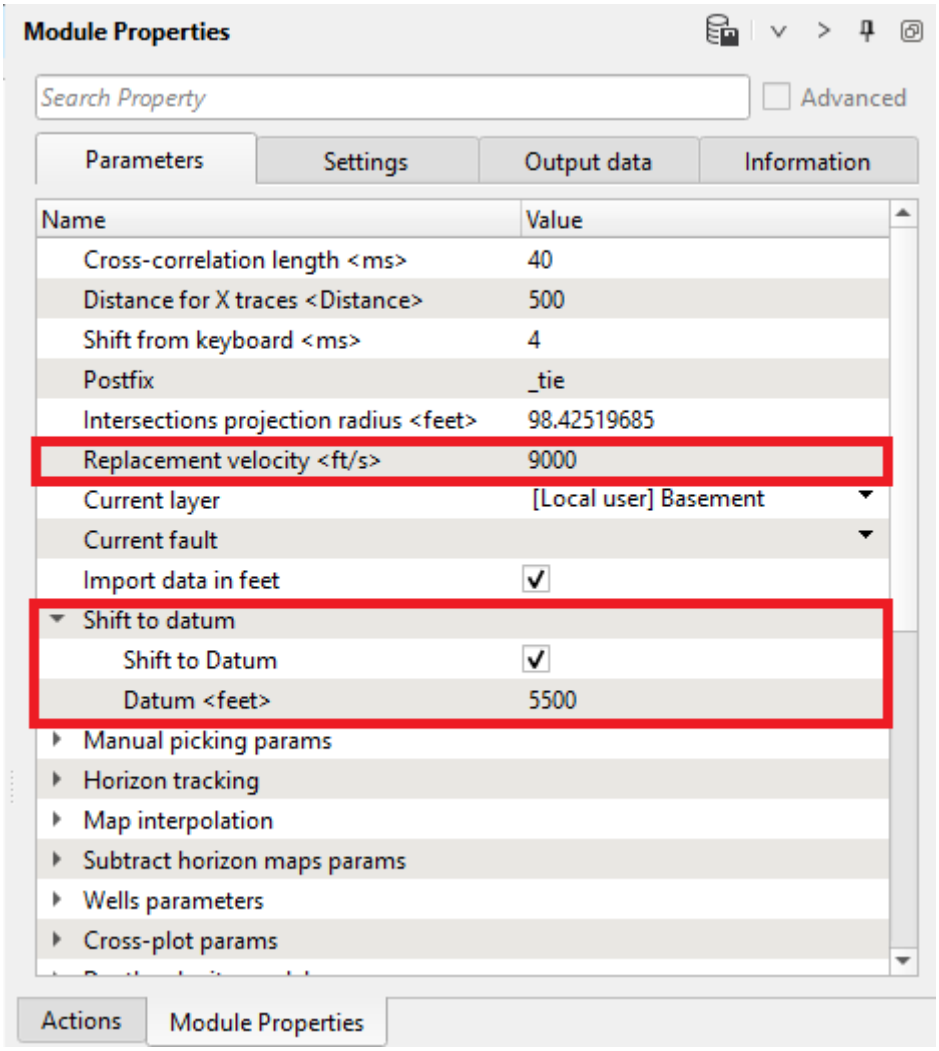
After confirming the file and import settings, choose **Time** or **Depth** in the **Select fault domain** dialog, then enter the input **Datum**.

Charisma faults can also be imported directly from the Data manager 124. In the **Faults** section, right-click the **Time** or **Depth** folder you want to import into and choose **Import fault** > **From Charisma...**. The imported faults are placed in that folder, so the **Select fault domain** dialog is not shown; you still choose the files, adjust the import settings and enter the input **Datum**.

For more information about fault formats supported by g-Space refer to Fault Data Formats 515

Before importing faults make sure that the project **datum** and **replacement velocity** are the same for importing faults otherwise in must be changed. To do this go to Module Properties 197 panel --> Parameters tab

© 2026 Geomage 2003 LTD



5.5 Import Shapefile

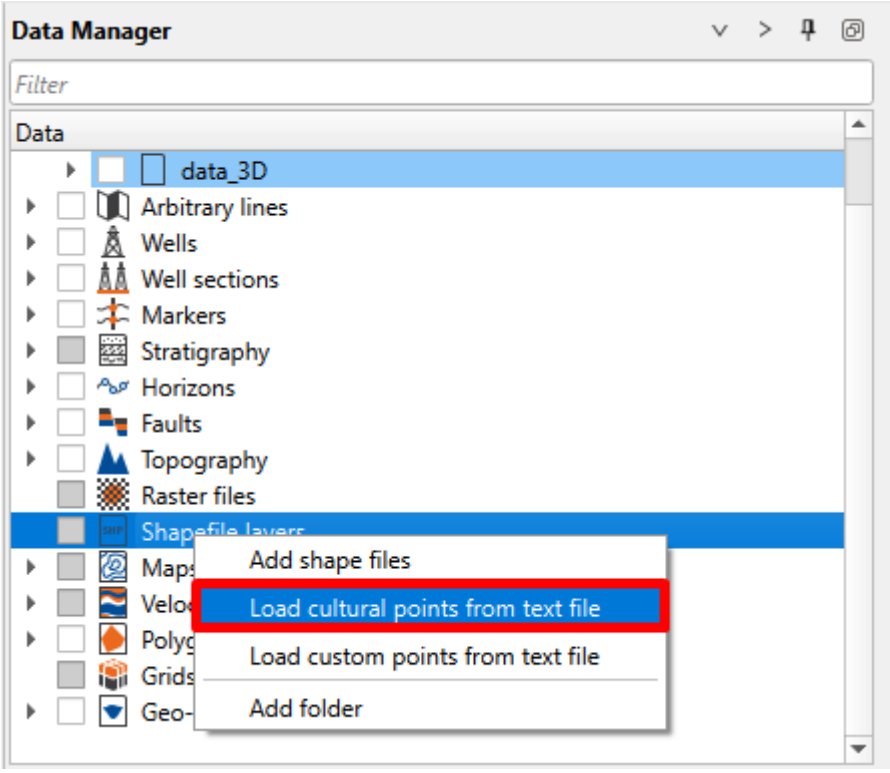
There are two ways to import shapefile into the project

Option 1: Open the Actions 214 panel:

press **Load shape files** button

Option 2: On **Shapefile layers** icon in Data manager 124 press right mouse button and choose **Add shape files** option

© 2026 Geomage 2003 LTD



Also you can import point attribute data by choosing Load cultural points from text file.

For more information about shapefiles format refer to GIS Data Formats 517

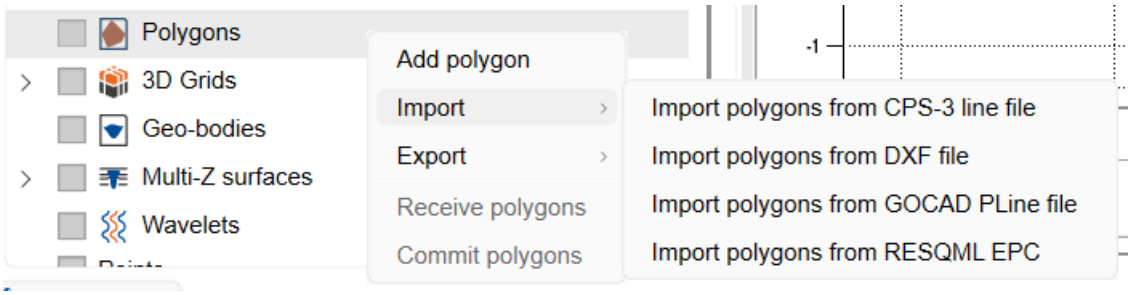
5.6 Import polygons

Polygons define closed 2D areas used to constrain maps, restrict volume calculations or delimit project extent. g-Space imports polygons from text files and stores them under **Polygons** in the Data manager 124.

How to import

1. Open Data manager 124.
2. Right-click the **Polygons** icon.
3. Point to **Import** and choose **Import polygons from CPS-3 line file**, then select the source file.
4. Confirm the coordinate reference system in the preview and press OK. The imported polygons appear as children of the Polygons node.

© 2026 Geomage 2003 LTD



Supported formats

- **CPS-3 line (.cps3)** — text-based line format; the same reader is used for CPS-3 map imports. Right-click **Polygons**, point to **Import** and choose **Import polygons from CPS-3 line file**.
- **GOCAD PLine (.pl)** — polylines / polygons from GOCAD. Right-click **Polygons**, point to **Import** and choose **Import polygons from GOCAD PLine file**.
- **DXF (.dxf)** — AutoCAD polygons and geobodies. See Import DXF 458.

For a description of the supported file layout, see GIS Data Formats 517, GOCAD Data Formats 519 and Model Data Formats 522.

See Also

- Export polygons 491
- GIS Data Formats 517
- Data manager 124

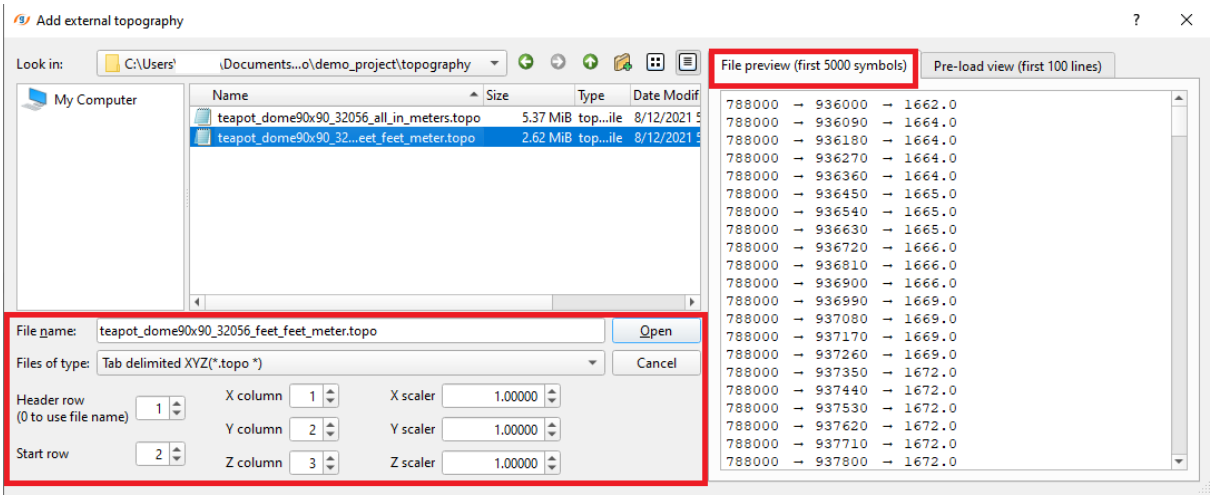
5.7 Import topography

In the Data Manager 124, expand **Topography**, right-click the external topography item, and choose **Load from file**. See Topography 160 for the available context-menu actions.

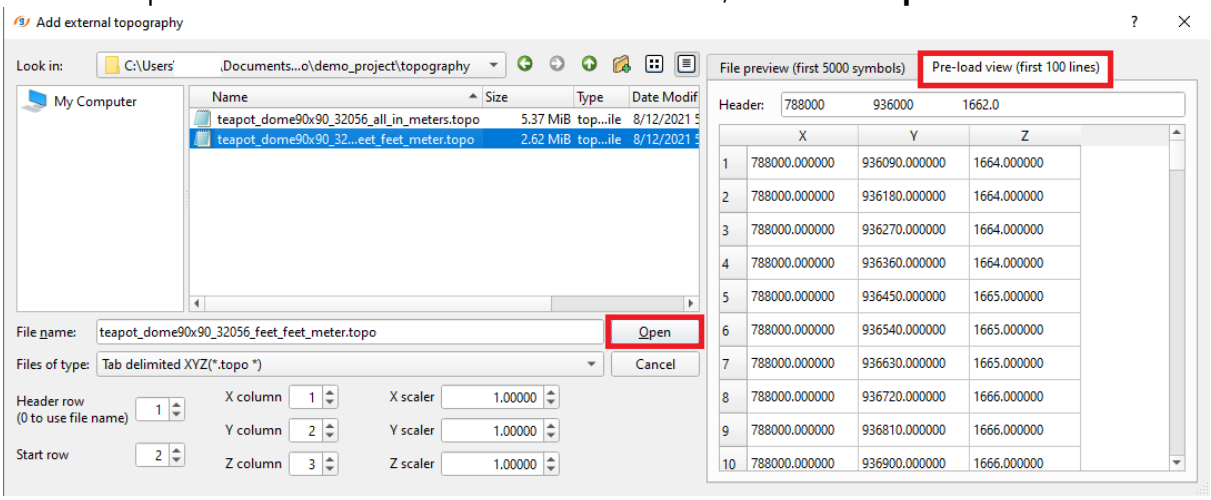
In the **Add external topography** dialog, select the source file and set the loading parameters:

- File type
- Headers row
- X,Y,Z columns and scale ratio

© 2026 Geomage 2003 LTD



Review the parsed coordinates on the **Pre-load view** tab, then select **Open**.



For more information about external topography formats supported by g-Space refer to Grid Data Formats 514

Downloading topography from the web

If you do not have a topography file, g-Space can download a digital elevation model (DEM) for the project area from an online elevation service. The downloaded DEM is stored as a georeferenced topography surface and can be used in processes that need elevation (for example velocity model creation). You can also download satellite basemap imagery for the same area - see Import satellite imagery 449.

5.8 Import images

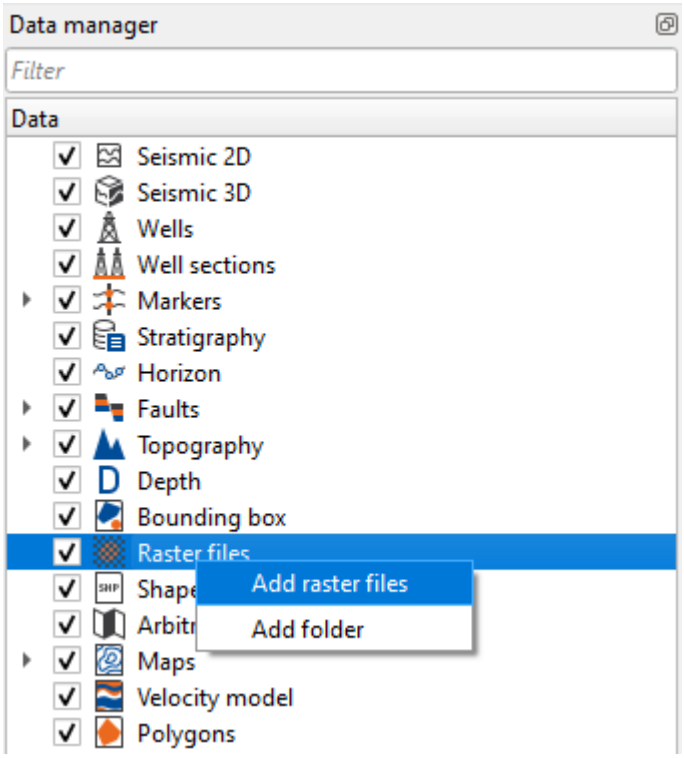
There are two ways to import image file into the project

© 2026 Geomage 2003 LTD

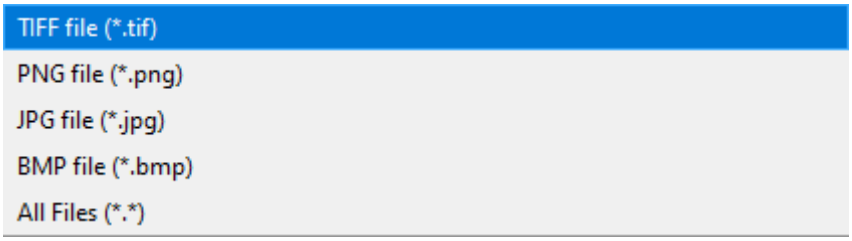
Option 1: Open the Actions 214 panel

press Load picture files button

Option 2: On **Raster files** icon in Data manager 124 press right mouse button and choose **Add raster files** option

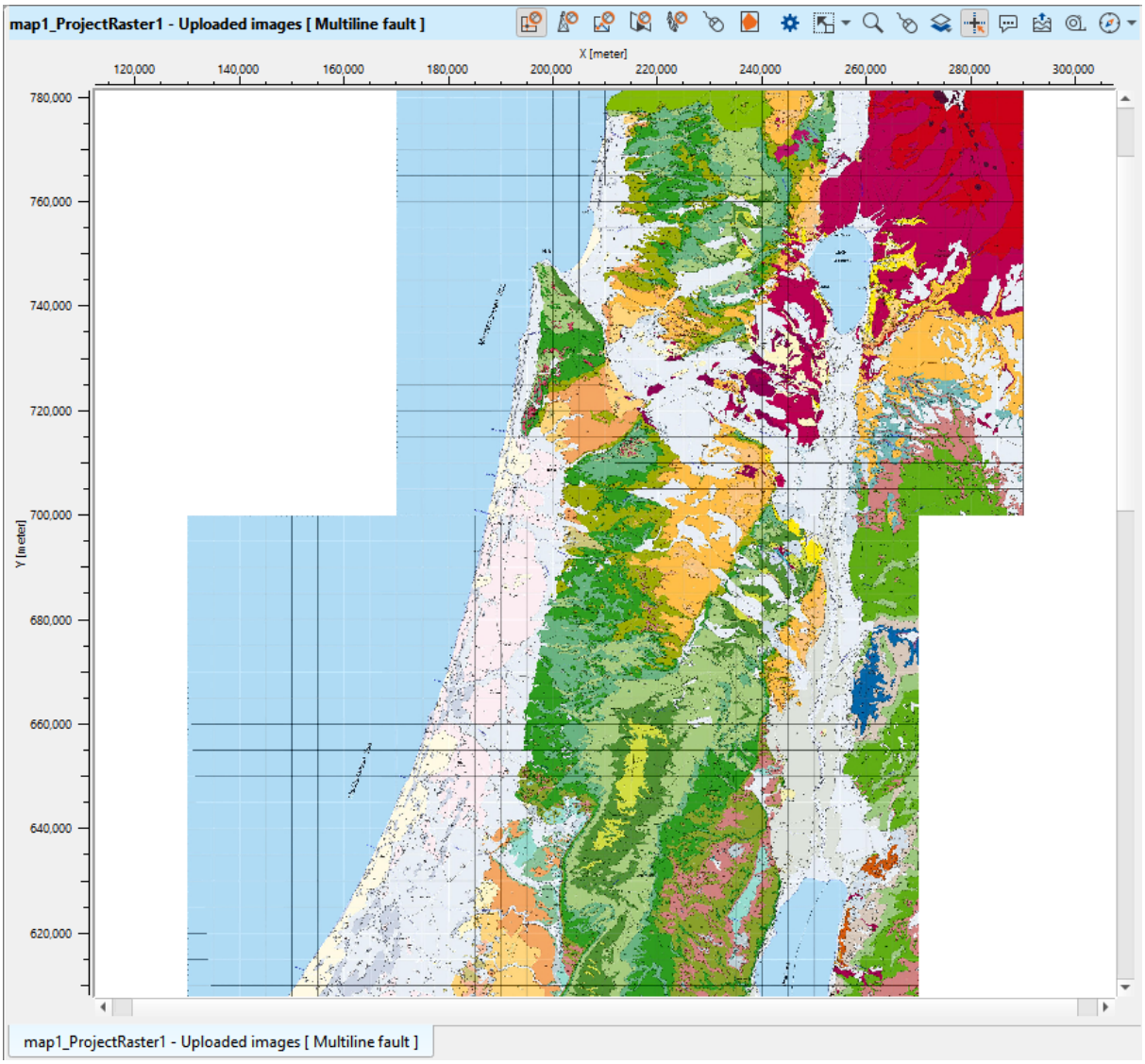


Locate file and press Open. You can load geo-referenced tiff, png, jpg, bmp formats



Loaded geo-referenced image on location map 235 view

© 2026 Geomage 2003 LTD



Note

The word file must have same name as image file and located in the same directory

Name	Size	Type	Date Mod
map1_ProjectRaster1.jgw	91 bytes	jgw File	6/28/2020
map1_ProjectRaster1.jpg	1.45 MiB	jpg File	6/28/2020
map2_ProjectRaster1.jgw	91 bytes	jgw File	6/28/2020
map2_ProjectRaster1.jpg	1.79 MiB	jpg File	6/28/2020
map3_ProjectRaster1.jgw	91 bytes	jgw File	6/28/2020
map3_ProjectRaster1.jpg	2.33 MiB	jpg File	6/28/2020
map4_ProjectRaster1.jgw	91 bytes	jgw File	6/28/2020
map4_ProjectRaster1.jpg	1.55 MiB	jpg File	6/28/2020

For more information about geo-referenced formats see Image Data Formats 516

© 2026 Geomage 2003 LTD

5.8.1 Import satellite imagery

Satellite imagery is downloaded on demand from a web map-tile service and saved as a georeferenced **GeoTIFF** (`.tif`). g-Space downloads the tiles that cover your project area, stitches them into a single image and stores it so it can be displayed as a cultural background under your data. This topic covers the download wizard; the result is imported as an image only.

Purpose

Fetch basemap / satellite imagery over a chosen rectangular area and store it as a georeferenced raster. The image is intended as a cultural background for maps and base-map views - it does not carry any seismic, well or horizon data. This is an import-only workflow: there is no matching export back to a tile service.

Launching the download

Open the **Satellite imagery downloader** wizard. The dialog opens with the project coordinate reference system already applied, so the area you enter is interpreted in project coordinates.

Step-by-step

1. Choose the area. Either pick a saved outline from the **Polygon** list, or leave it on **User defined** and type the bounding box into **Min X**, **Max X**, **Min Y** and **Max Y**. Coordinates are shown in metres (or feet, following the active measurement system).
2. Set the **Zoom level** (1-19; 15 by default). A higher zoom gives finer detail but downloads many more tiles.
3. Check the **Description** panel: it reports the CRS, the estimated number of **Tiles** and the resulting **Image size** in pixels for the current area and zoom.
4. Press **Download**. The wizard converts the area to geographic coordinates (WGS84), downloads each map tile with a progress bar, and stitches them into one image.
5. When the download finishes the georeferenced `.tif` is written to the project folder and can be added as a background image.

© 2026 Geomage 2003 LTD

Notes

- The output is a 3-band RGB GeoTIFF georeferenced in WGS84 (`EPSG:4326`), saved into the project directory with an automatic name such as `satellite_z15_YYYYMMDD_HHMMSS.tif`.
- Imagery is fetched from an online map-tile service, so an internet connection is required. Tiles are 256 x 256 pixels and are combined into a single stitched raster.
- Very large requests take a long time; the wizard warns before downloading if the area is unusually big (thousands of tiles), and extremely large images are rejected. Reduce the area or lower the zoom level if needed.
- The download runs in the background and can be stopped with Cancel while it is in progress.

See Also

Image Data Formats 516 | Import images 446

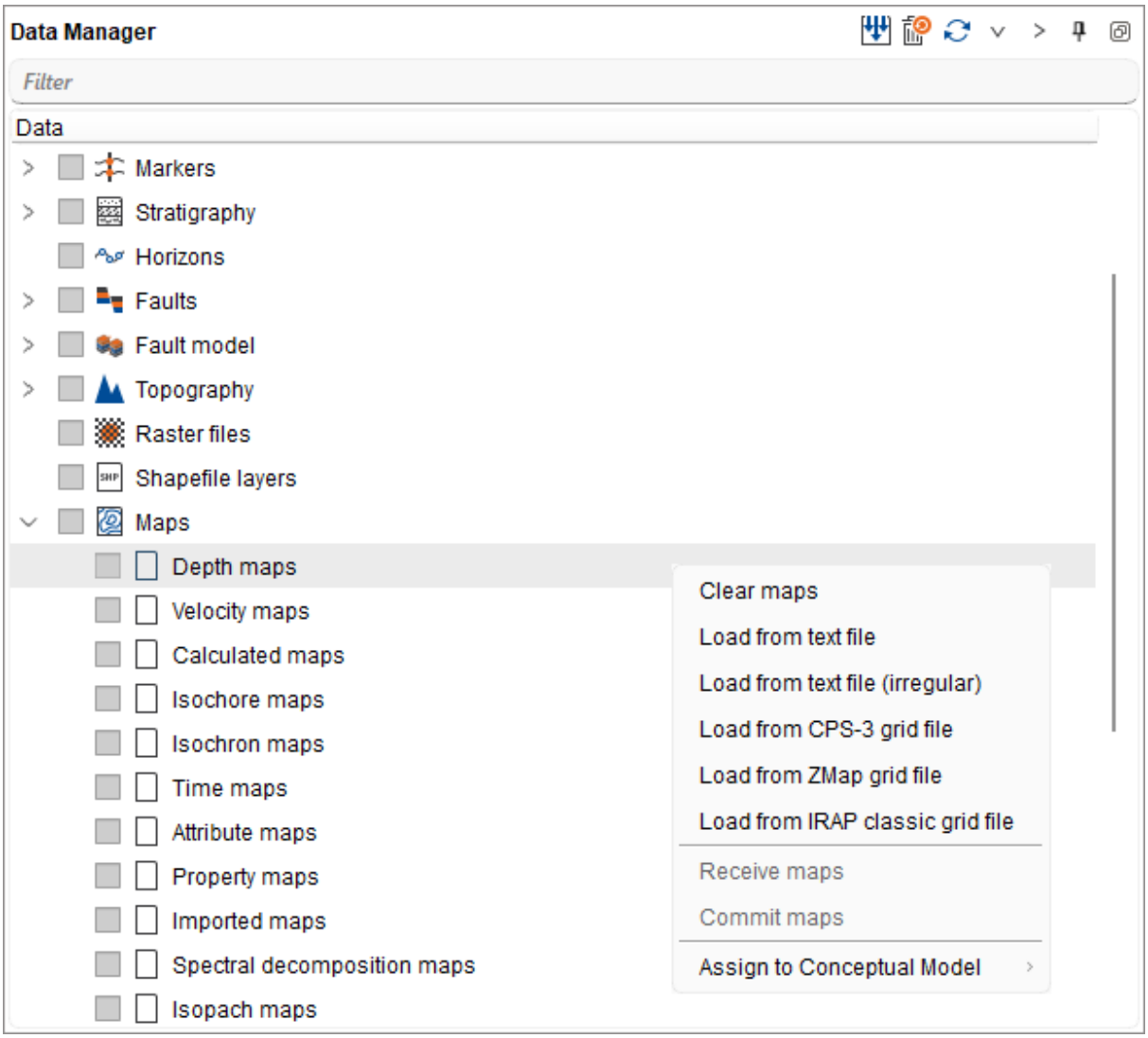
5.9 Import maps

There are five ways to import maps files into the project:

1. Load from text file
2. Load from text file (irregular)
3. Load from CPS-3 grid file
4. Load from ZMap grid file
5. Load from IRAP classic grid file (*.irap / *.gri / *.txt)

In Data manager 124 in Maps icon decide in which set of maps data will be loaded. Press right mouse button and choose one of the load commands:

© 2026 Geomage 2003 LTD



The CPS-3, ZMap and IRAP classic grid loaders let you select **several files at once** in the load dialog. Each selected file is imported as a separate map into the chosen set of maps. The plain text-file loaders load one file at a time.

An imported map is named after the file it comes from: g-Space uses the file name without its extension as the map name in Data manager. For example, the file `Map_ConInt.cps3` gives a map called `Map_ConInt`. For CPS-3 files the grid name written in the file header is not used. When you load a single file, the **Set name of the loaded map** dialog opens with this name already suggested in **Loaded map name**, so you can keep it or type another one. When several files are selected at once, no name is asked and each map keeps the name of its own file. The **Load from text file (irregular)** import wizard suggests the same name in its **Map name** field.

© 2026 Geomage 2003 LTD

If the chosen set of maps already contains a map with the same name, a number is added to the new name to keep it unique, for example `Map_ConInt (1)`.

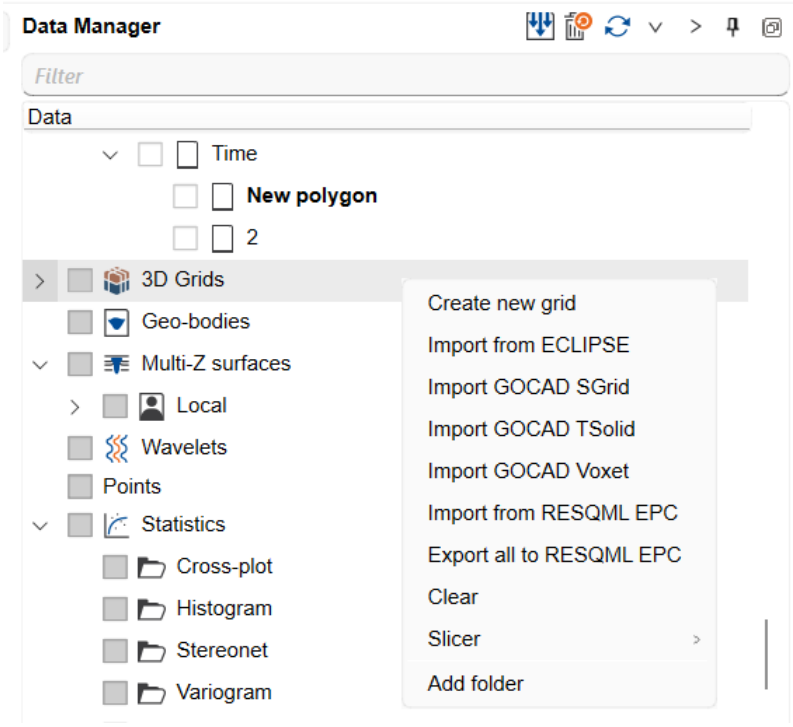
For more information about grid formats supported by g-Space refer to Maps Data Formats 514

5.10 Import grids

This topic describes how to import a grid stored in **Eclipse format (.dat)**. The same **3D Grids** menu also imports GOCAD SGrid, GOCAD TSolid, GOCAD Voxet and RESQML EPC grids, which are described in Model Data Formats 522.

To import an Eclipse grid into **g-Space**, follow these steps:

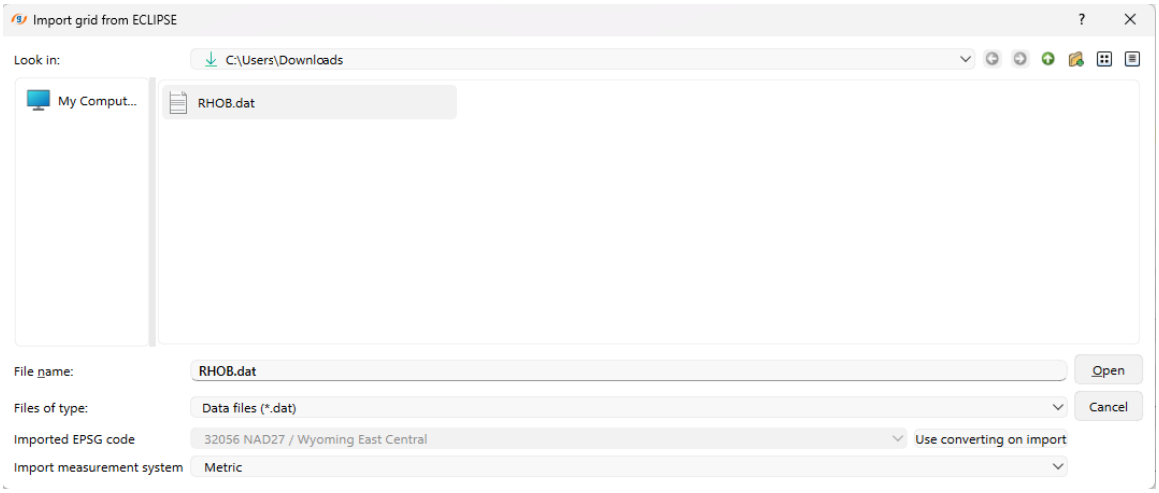
1. Open the **Data Manager**.
2. Navigate to the **3D Grids** folder.
3. Click **Import from ECLIPSE**.



4. A pop-up window will appear.
5. Select the **.dat file** that contains the Eclipse grid data.

© 2026 Geomage 2003 LTD

6. Select the coordinate system of the file in the **Imported EPSG code** row, and select **Use converting on import** if the file coordinates must be converted to the project coordinate system.
7. Click **Open** to complete the import.



If the file declares its own unit system (**METRIC** or **FIELD**), g-Space applies that declaration and sets **Import measurement system** accordingly. The value selected in the dialog is used only for files that do not declare a unit system. Grids exported from g-Space always declare their unit system, so they are imported back at their original position.

The imported grid will appear in the **3D Grids** folder and can be visualized and edited in g-Space.

5.10.1 Import 3D models

g-Space imports 3D geological models stored in the **GOCAD** / SKUA-GOCAD ASCII exchange formats. This topic covers loading a GOCAD **SGrid** (`.sg`), a GOCAD **TSolid** (`.so`) and a GOCAD **Voxet** (`.vo`) into a g-Space project as grids.

Purpose

These GOCAD formats carry 3D grid geometry together with node- or cell-based scalar properties. Importing them brings an externally built structured grid, tetrahedral mesh or voxel volume into the g-Space grid store so it can be viewed, sampled and used alongside native

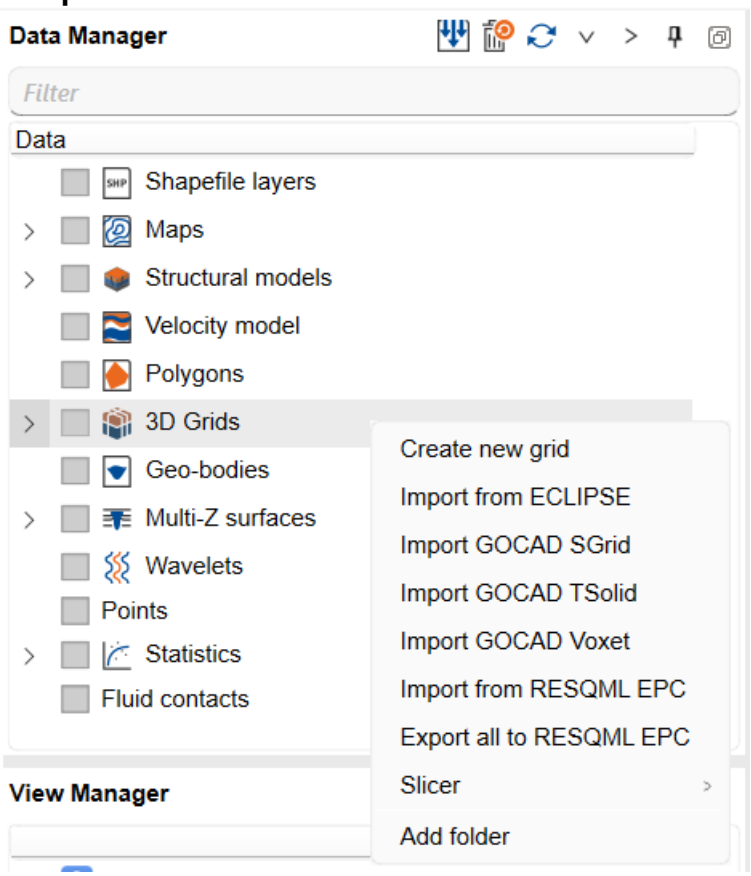
© 2026 Geomage 2003 LTD

grids. For a TSolid or a Voxet, the time or depth domain is read from the file metadata; an imported SGrid is always created in the depth domain. The Z sign convention (Z-down depth versus Z-up elevation) is read from the file metadata for all three formats, and elevation values are negated so that Z is stored as depth, positive downwards.

Launching the import

In the Data manager, right-click the **3D Grids** root node and choose one of the GOCAD import actions:

- **Import GOCAD SGrid**
- **Import GOCAD TSolid**
- **Import GOCAD Voxet**



Supported formats

- **GOCAD SGrid** (`.sg`) — a 3D structured (i, j, k) grid. The `.sg` file is an ASCII header holding the grid dimensions, coordinate-system and unit metadata and the property list; it references a companion ASCII data file that holds the per-node X / Y / Z coordinates and property columns, so both files are needed. A grid whose nodes form a regular rotated-rectangle

© 2026 Geomage 2003 LTD

lattice loads as a g-Space S-Grid; a warped (faulted or curved) grid is imported as a tetrahedral grid. Properties may be point- or cell-aligned.

- **GOCAD TSolid** (`.so`) — an unstructured tetrahedral mesh: a list of vertices and tetrahedra, optionally grouped into TVOLUME parts, with scalar properties stored per cell or on the vertices. It is imported as a g-Space tetrahedral grid, and each property keeps its original name. A property that only carries vertex (node) values is imported as well: each tetrahedron takes the average of its four vertex values, so those values arrive as per-cell averages rather than exact node values; missing or no-data vertex values are skipped, and a tetrahedron with no usable vertex value stays undefined. A single file may contain several TSolid objects, and multiple parts within one object are merged.
- **GOCAD Voxet** (`.vo`) — a regular, axis-oriented 3D voxel grid defined by an origin, three axis vectors and a sample count per axis, with property values kept in external property files (ASCII text or raw binary IEEE floats). Because the sampling is regular, g-Space imports a Voxet as a structured S-Grid. Voxet is import only.

Step-by-step

1. Right-click the **3D Grids** root in the Data manager and pick the action matching your file: **Import GOCAD SGrid**, **Import GOCAD TSolid** or **Import GOCAD Voxet**.
2. In the file dialog, select the GOCAD header file (`.sg`, `.so` or `.vo`).
3. g-Space reads the header and its referenced data / property files. For an SGrid or Voxet, keep the companion data and property files next to the header. If an SGrid data file cannot be found you are prompted to browse for it; a Voxet property file that cannot be found is reported as a warning and that property is skipped.
4. The grid, its dimensions, domain and properties are created from the file, and the new grid appears under the **3D Grids** node ready to view and query.

Notes

- An SGrid header only points at the geometry, so its companion ASCII data file must travel with it; a regular-looking header can still reference a warped points file, so the actual node coordinates decide whether the grid becomes an S-Grid or a tetrahedral grid.
- For a TSolid or a Voxet, the time or depth domain is taken from the axis name / unit metadata. An imported SGrid is always created in the **Depth** domain, whatever its axis metadata says, so a time-domain `.sg` file still arrives as a depth-domain grid. Property no-data values are honoured and mapped to undefined nodes.
- The Z sign convention (ZPOSITIVE Depth versus Elevation) is taken from the header, and elevation values are negated so that Z is stored as depth, positive downwards. A `.sg` header with no ZPOSITIVE record is read as depth-positive-down and its Z values are used

© 2026 Geomage 2003 LTD

unchanged, so a shared file that actually holds elevations but omits that record loads with its Z values inverted. If an imported SGrid appears inverted in the 3D view, check whether its header declares ZPOSITIVE Elevation.
- Voxet is supported for import only. Export back to GOCAD is available for structured grids (SGrid) and tetrahedral grids (TSolid).

See Also

Model Data Formats 522 | Export 3D models 485 | GOCAD Data Formats 519

5.10.2 Import RESQML

RESQML is the Energistics standard for exchanging reservoir and geological models. A RESQML model is delivered as an EPC package - a single `.epc` file that references its bulk array data in linked `.h5` files. g-Space imports RESQML EPC packages directly from the Data manager 124.

Purpose

Load geological objects stored in a RESQML v2.0.1 EPC package into a g-Space project. A single package can carry several object types, each of which maps to the matching g-Space data category:
- **Polylines / polygons** (PolylineSetRepresentation) into **polygons**.
- **Point sets** (PointSetRepresentation) into **point sets**.
- **Multi-Z surfaces** (TriangulatedSetRepresentation) into free **Multi-Z surfaces**.
- **Tetrahedral / 3D grids** (UnstructuredGridRepresentation) into tetrahedral **grids**, including any cell properties they carry.
- **IJK / corner-point grids** (IjkGridRepresentation) into tetrahedral **grids** as well: every grid cell is divided into six tetrahedra, and the cell properties are carried over to them.

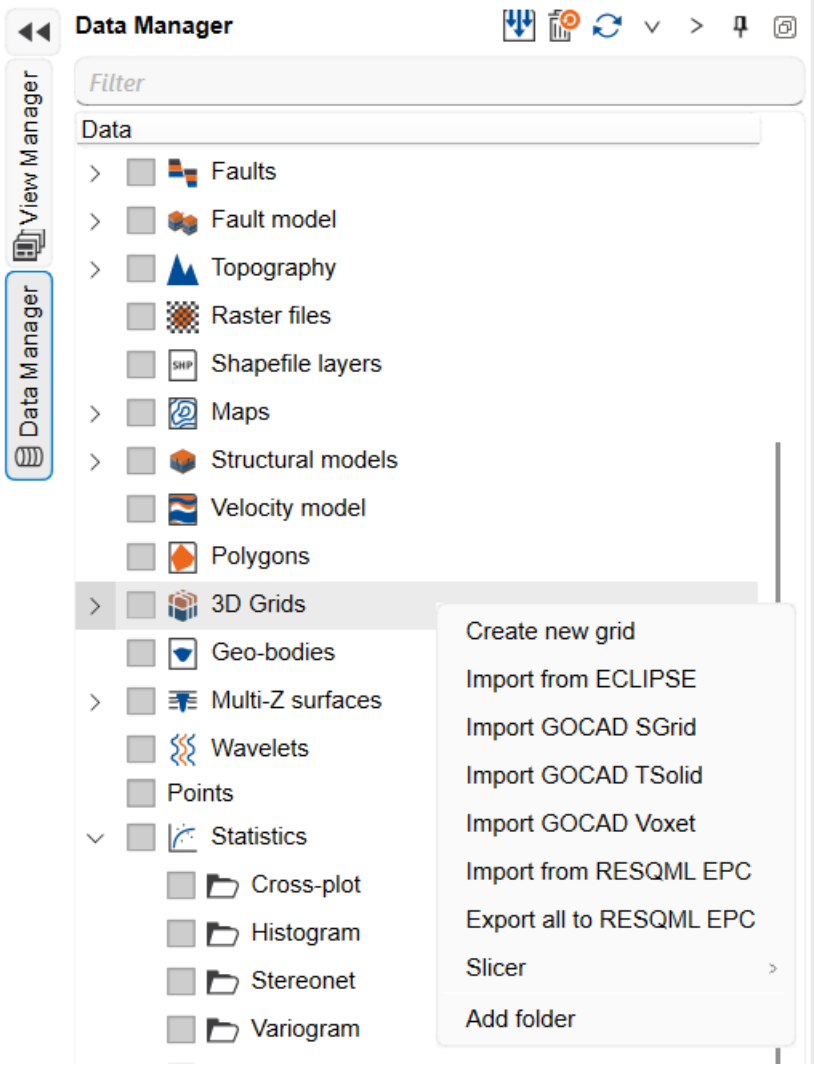
Coordinates are read from the package CRS and the vertical axis is converted to the g-Space depth-positive convention automatically, so no axis flipping is required afterwards.

Launching the import

RESQML import is launched per object type from the Data manager 124 tree. Right-click the branch that matches the data you want to load and choose the RESQML action:
- **Polygons** root -> **Import** -> **Import polygons from RESQML EPC**.
- **Points** root -> **Import from RESQML EPC...**.

© 2026 Geomage 2003 LTD

- **Multi-Z surfaces** Time or Depth branch -> **Import from RESQML EPC** (the domain is taken from the branch you click).
- **3D Grids** root -> **Import from RESQML EPC**.



Step-by-step

1. In the Data manager, right-click the target branch and choose the matching **Import ... from RESQML EPC** action.
2. In the file dialog set the filter to **RESQML EPC (*.epc)** and select one or more `.epc` packages. Keep each package next to its linked `.h5` array files.
3. g-Space reads every representation of the requested type from the package; each representation becomes one g-Space object, with all of its geometry patches merged.
4. For grids, confirm the import preview: it lists each detected grid with its RESQML type, the number of source cells and the number of tetrahedra they produce, and how many cell properties were found. Confirm it to import the listed grids, or decline to skip that package.

© 2026 Geomage 2003 LTD

5. Review the import report: it lists how many objects were read, the RESQML version, and any non-fatal warnings (for example skipped or empty representations). Non-tetrahedral cells in an unstructured grid are skipped with a warning.
6. The new objects appear under the branch you imported into and are drawn in the associated views.

Notes

- g-Space reads RESQML v2.0.1 packages. The `.epc` file is only a container - its bulk arrays live in the linked `.h5` files, so copy or move them together.
- The RESQML EPC package carries no time/depth domain marker, so for Multi-Z surfaces the destination domain is defined by the Time or Depth branch you import into rather than read from the file.
- Unstructured grids are imported as tetrahedral meshes: only 4-node (tetrahedral) cells are kept, and other cell shapes are skipped non-fatally.
- IJK / corner-point grids are converted to tetrahedral grids too, so the result is a tetrahedral grid rather than a native IJK grid. Cells whose geometry is not defined in the package are skipped and counted in the import report.

See Also

Model Data Formats 522 | Export RESQML 486

5.10.3 Import DXF

DXF (Drawing Exchange Format) is the AutoCAD interchange format. g-Space imports `.dxf` files that carry polygon outlines and geobody (3D surface) geometry. Reading is handled internally by the bundled `libdxfrw` library, so no external CAD software is required.

Purpose

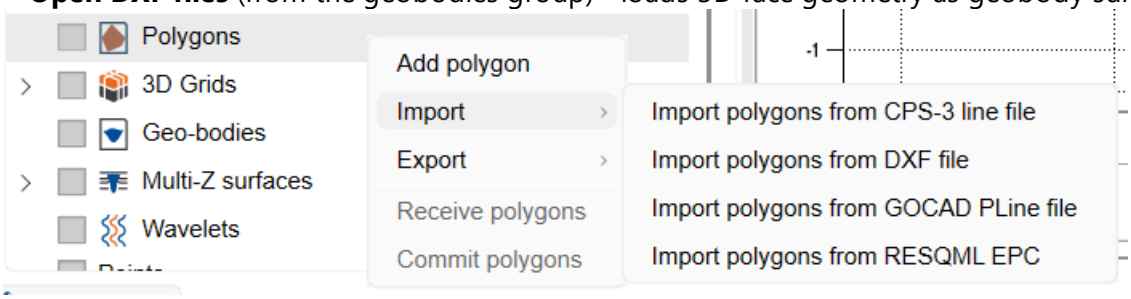
Bring geometry authored in CAD tools into a g-Space project. Two kinds of objects are read from DXF: closed polylines are imported as *polygons*, and 3D face meshes are imported as *geobodies* (free 3D surfaces). Import is read-only and requires a Seismic Interpretation license.

Launching the import

© 2026 Geomage 2003 LTD

Both importers are launched from the Data manager context menu. Right-click the target group and choose:

- **Import** > **Import polygons from DXF file** - loads closed polylines into the polygons group.
- **Open DXF files** (from the geobodies group) - loads 3D face geometry as geobody surfaces.



Step-by-step

1. In the file dialog, keep the filter on **DXF files (*.dxf)** and select one or more `.dxf` files.
2. For polygon import, set the coordinate reference system and measurement system in the dialog's additional options. DXF files carry no coordinate-system metadata, so confirm the file coordinates and units match the project.
3. For geobody import, tick **Invert Z axis** if the DXF Z values should be flipped to match the project's depth convention.
4. For polygons, choose which DXF layers to import in the **Select DXF layers to import** dialog. All polylines on the selected layers are imported; `TEXT` labels sitting inside a closed polyline supply its name.
5. Confirm to load. Imported polygons appear in the polygons group and imported geobodies in the geobodies group of the Data manager.

Notes

- Polygon import reads `LWPOLYLINE` and `POLYLINE` entities; the closed flag on the entity sets whether the resulting polygon is closed.
- Geobody import reads `3DFACE` entities and stitches them into a single triangulated free surface per file.
- Polylines with no name text fall back to a `<layer>_<index>` name; geobodies are named after the file.
- DXF import does not store a coordinate system, so imported points are treated in the project CRS you select at import time.

© 2026 Geomage 2003 LTD

See Also

Model Data Formats 522 | Export DXF 489 | Import polygons 444

5.11 Import points

Point sets and cultural / custom shape points are imported from plain delimited text (CSV / TAB / `.txt`) using the shared column-mapping wizard, where you assign each column to a field and choose the target object type.

Purpose

Load scattered point data - point sets, cultural points or custom shape points - from a text file into the project. The first two columns hold the X and Y coordinates; the remaining columns are loaded as attributes.

Launching the import

In the Data manager 124, right-click the **Points** (or cultural data) node and choose the import command, then select the text file.

Step-by-step

1. Right-click the **Points** node in the Data manager and choose the import-from-text command.
2. Select the delimited text file (CSV / TAB / TXT).
3. In the *column-mapping wizard*, set the separator, header row and start row, then map columns to X, Y and the attribute fields.
4. Check the parsed preview - fields with the wrong type are highlighted - then confirm to load the points.

Notes

- The same column-mapping wizard is used for well headers, markers, inclinometry and other ASCII imports; only the target object type differs.

© 2026 Geomage 2003 LTD

- Cultural / custom shape points use the same text layout and are loaded the same way.

See Also

Point Data Formats 523 | Export points 499 | GIS Data Formats 517

5.12 Import from g-Platform

g-Platform is a companion database application that stores project data centrally. This topic covers importing seismic data held in a g-Platform database into the current g-Space project. Supported seismic files are `.gsd`, `.sgy` and `.segy`.

Purpose

Pull seismic volumes and lines from a shared g-Platform database into g-Space. Import first connects to the database, then lets you select the seismic files to bring into the project. Only seismic file types (`.gsd`, `.sgy`, `.segy`) are imported.

Connecting to g-Platform

Import starts with the **Connect to g-Platform database** dialog. g-Platform uses a PostgreSQL database, so enter the connection details and press **Test** to verify the settings before connecting.

- **Hostname** — database server address (default `localhost`).
- **Port** — PostgreSQL port (default `5432`).
- **Database**, **User** and **Password** — g-Platform database name and credentials (defaults `gplatform` / `gplatform`).
- **Change files path (Windows-Linux)** — optional. Enable it and fill in the **Remote path prefix** and **Local path prefix** to translate stored file paths between operating systems.

Press **Test** to check the connection (a **Connection successful** message confirms it), then press **Connect** to open the import dialog. All connection fields must be filled in before connecting.

Step-by-step

1. Open the **Connect to g-Platform database** dialog, enter the connection details, press **Test**, then press **Connect**.

© 2026 Geomage 2003 LTD

2. In the **Import data from g-Platform** dialog, browse the **Available objects** list on the left. Use the **Search files** box to filter by name.
3. Select one or more files and move them to the **Objects to import** list on the right with the **>** button (or **>>** to move all). Double-clicking a file moves it as well; the **<** and **<<** buttons move files back.
4. Files that are missing on disk are shown with a red background and cannot be selected. Files already present in the project are shown greyed out; clear **Show existing objects** to hide them.
5. Press **OK** to import the selected seismic files into the project, or **Cancel** to abort.

Notes

- The **Available objects** list shows the file name, file size and full path of each database file.
- Only seismic files are imported. Existing project files are marked as grey shadow items so you can see what is already loaded without re-importing it.
- If the g-Platform files were written on a different operating system, use **Change files path (Windows-Linux)** on the connection dialog to remap the path prefixes.

See Also

Export to g-Platform 503 | Import Seismic 373 | Seismic Data Formats 511

© 2026 Geomage 2003 LTD

Export data

6 Export data

This chapter will guide you through exporting different types of data from g-Space.

Here is the list of the data types you are able to export form g-Space:
- Well data - well headers 464, markers 468, checkshots 471, well logs 474, well inclinometry
- Seismic horizons 478
- Fault sticks 480
- Grid data 484
- Polygons 491

Selecting well data to export

When you export well data from the Data manager 124 — right-click the **Wells** folder or an individual well and choose **Export** — a setup window opens so you can choose exactly which fields are written. The **Export** submenu offers:
- **Wellheads** - opens the **Wellheads Export Setup** window (UWI, Well Name, Company Name, Type, Status, X (Easting), Y (Northing), Completion Date, Total Depth, Kelly Bushing, Ground Level, CRS, Comments).
- **Inclinometry** - opens the **Inclinometry Export Setup** window (Well Name, UWI, MD, Inclination, Azimuth, TVD, TVDSS, X, Y, CRS).
- **Checkshot** - opens the **Checkshot Export Setup** window (Well Name, UWI, MD, TVD, TVDSS, OWT, TWT, Velocity, Average Velocity, CRS).
- **LAS** - opens the **LAS Export Setup** window, with **LAS format (*.las)** and **Excel format (*.xlsx)** outputs; here you also pick which log curves to include.

Each setup window lists the available fields as check boxes with **Select all** and **Deselect all** buttons; only the ticked fields are written to the exported file.

For more information about supporting formats please refer to chapter Formats 506

6.1 Export well headers

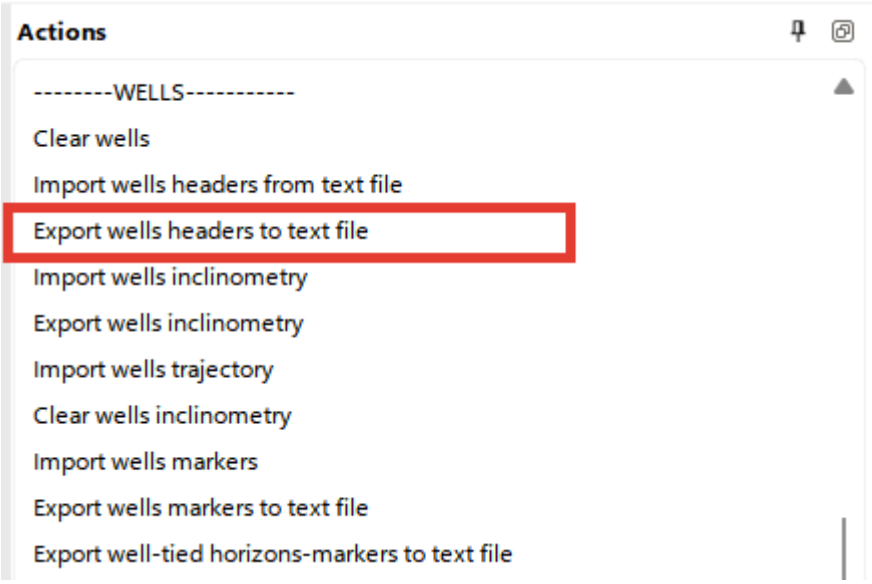
To export well headers, right-click the **Wells** folder or an individual well in Data Manager 124, choose **Export**, and then choose **Wellheads**. In the **Wellheads Export Setup** window, select the fields to include and click **OK**.

Choose the output folder and file name, then click **Save**.

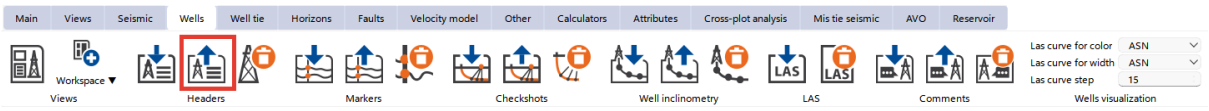
For more information about well headers format please refer to the chapter Well Data Formats 507

© 2026 Geomage 2003 LTD

Export well data available from Actions panel 123 as well



And from the Wells bar 46 from Headers section



Exported Well Headers Content

The exported file will contain the following well header information by default:

- UWI
- Well Name
- Company Name
- Type
- Status
- X (Easting)
- Y (Northing)
- Completion Date
- Total Depth
- Kelly Bushing
- Ground Level
- Coordinate Reference System (CRS)
- Comments/Annotations

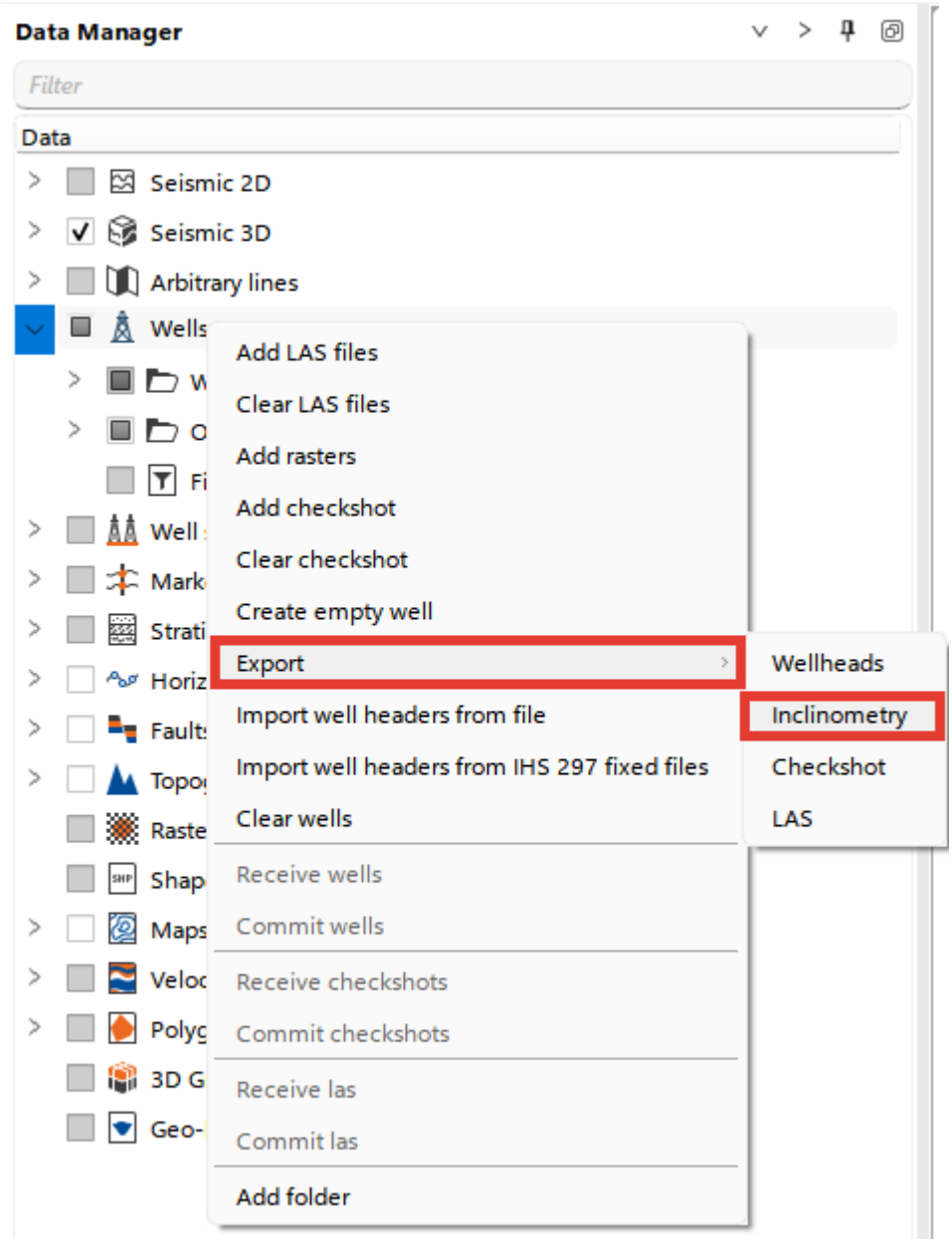
© 2026 Geomage 2003 LTD

6.2 Export inclinometry

To export well inclinometry press Right Click Mouse on Wells icon in Data Manager 124 and choose Export data and then **Inclinometry**

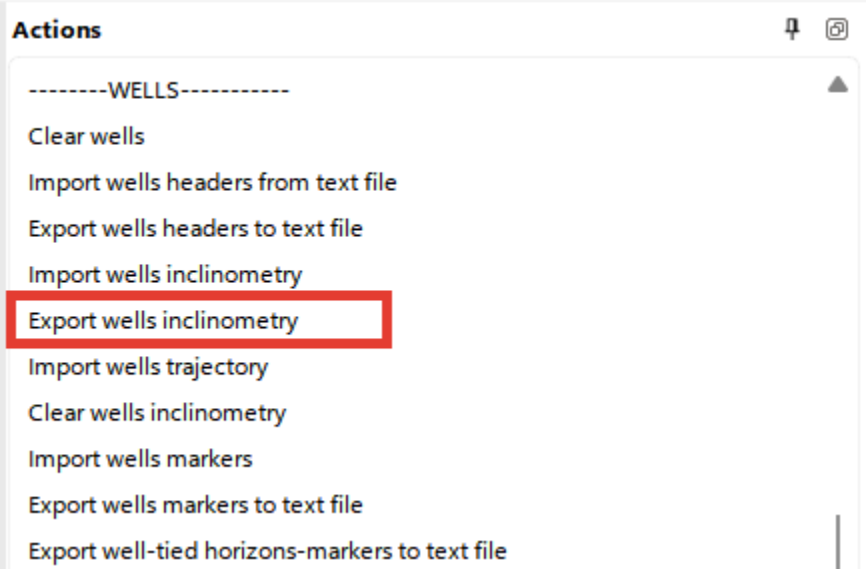
Locate file directory, choose file format and click **Save**

For more information about well headers format please refer to the chapter Well Data Formats 507



© 2026 Geomage 2003 LTD

Export well data available from Actions panel 123 as well



And from the Wells bar 46 from Well inclinometry section

Exported Inclinometry Content

The exported file will contain the following inclinometry data by default:

- Well Name
- UWI
- Measured Depth (MD)
- Inclination (INCL)
- Azimuth (AZIM)
- True Vertical Depth (TVD)
- True Vertical Depth Subsea (TVDSS)
- Easting (X)
- Northing (Y)
- Coordinate Reference System (CRS)

The MD, Inclination and Azimuth values come from the loaded well survey, while TVD, TVDSS, Easting (X) and Northing (Y) are values calculated by g-Space from that survey.

For an Imperial project, the exported MD, TVD and TVDSS values are written in feet. The corresponding column headers are MD (ft), TVD (ft) and TVDSS (ft).

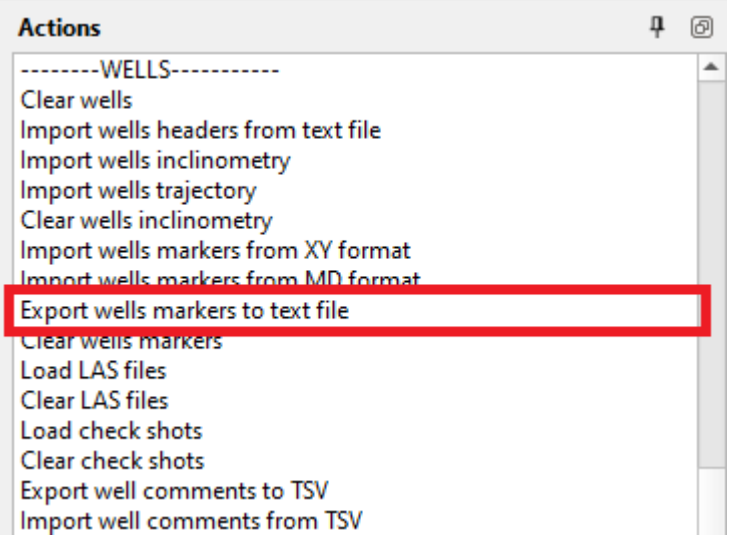
© 2026 Geomage 2003 LTD

During export g-Space opens a field-selection step where you choose which of these columns to include in the output file before saving.

6.3 Export markers

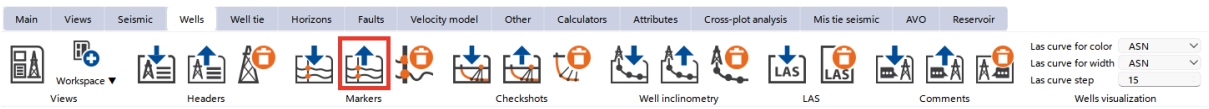
There are two options for exporting markers: **Export Wells Markers to Text File** for exporting standard well markers, and **Export Well-Tied Horizons-Markers to Text File** for exporting markers after performing a Well tie 567.

In Action panel 214 choose Export wells markers to text file



Locate file directory and click **Save**

It's also an option to export markers from Wells bar 46



For more information about well markers format please refer to chapter Well Data Formats 507

Markers Export from the Data Manager

© 2026 Geomage 2003 LTD

You can also export a marker set directly from the Data Manager. Right-click the marker set and choose **Markers Export**.

This opens the **Markers Export Setup** wizard, which follows the same style as the Well Headers and Well Inclinometry export wizards. Under **Select fields to export:** tick the marker fields/columns you want to include, then click **OK** and choose the output file. Use **Select All** or **Deselect All** to check or clear every field at once.

Any custom marker attributes defined on the marker set are listed as additional fields in the wizard, so they can be exported as well. They are ticked when the wizard opens, just like the standard fields, and each ticked attribute is written as an extra column after the standard columns, in alphabetical order. Custom attribute columns are added in the same way when you use **Export wells markers to text file**.

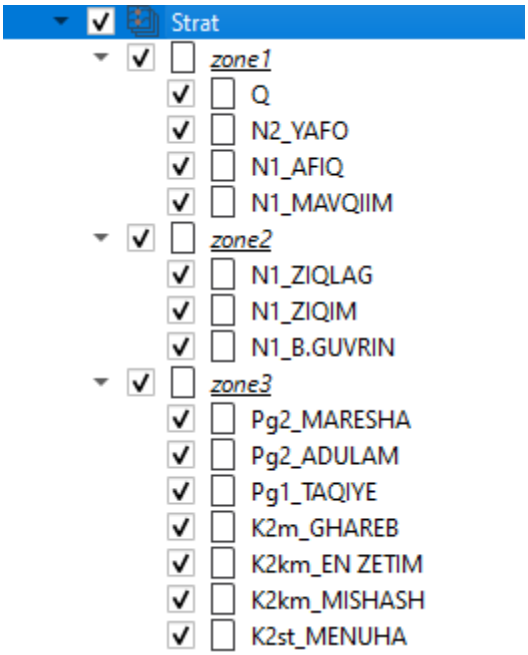
The exported file will contain the following standard marker information by default:

- **UWI**: Unique Well Identifier
- **Wellname:** Name of the Well
- **Marker:** Marker Name or Identifier
- **X:** Easting (X coordinate)
- **Y:** Northing (Y coordinate)
- **MD:** Measured Depth
- **Age:** Age of the Formation
- **TVD:** True Vertical Depth
- **TVDSS:** True Vertical Depth Subsea
- **TWT_CRD:** Two-Way Travel Time (Corrected Datum)
- **TWT_SRD:** Two-Way Travel Time (Sea Reference Datum)
- **Type:** Type of Marker
- **Unconformity:** Indicates Unconformity Type or Status

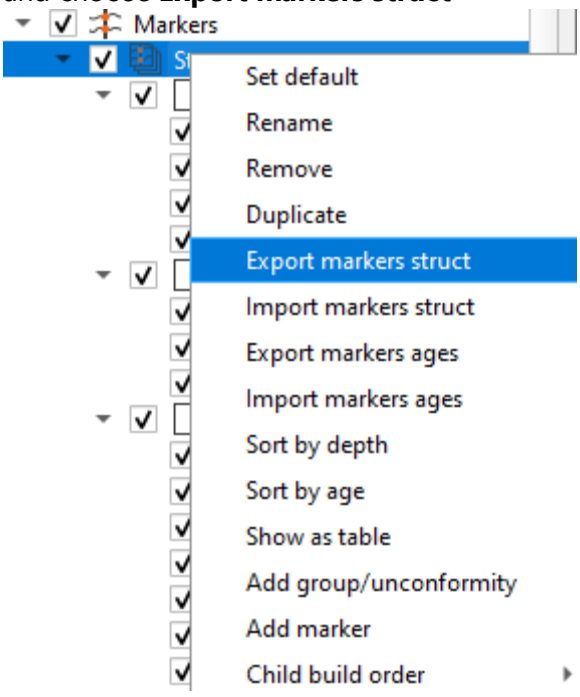
6.4 Export markers structure

Sometimes markers need to be organized with different levels of hierarchy (see example below)

© 2026 Geomage 2003 LTD



To export markers structure press Right Click Mouse on Markers set icon in Data Manager 124 and choose **Export markers struct**



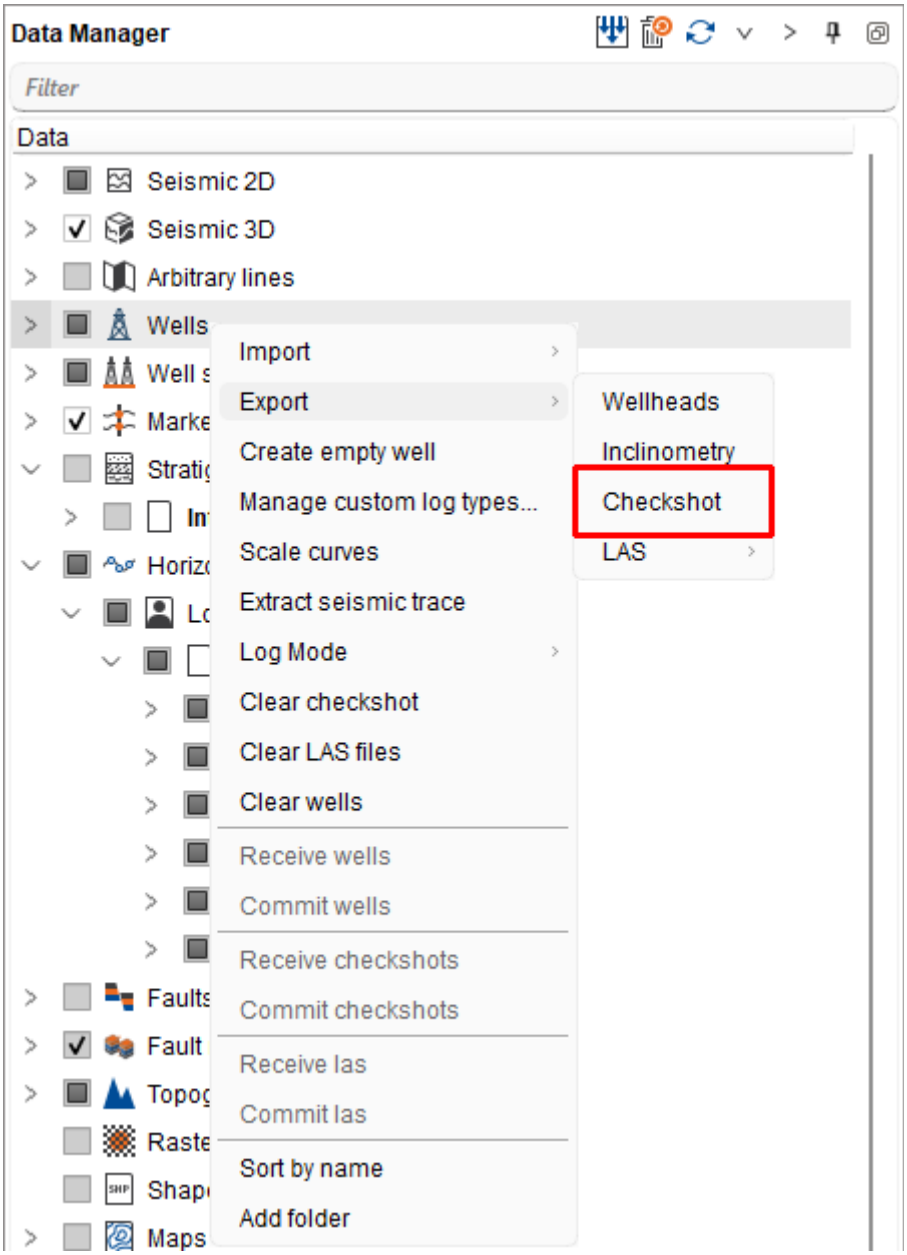
For more information about well markers structure format please refer to chapter Well Data Formats 507

© 2026 Geomage 2003 LTD

6.5 Export checkshots

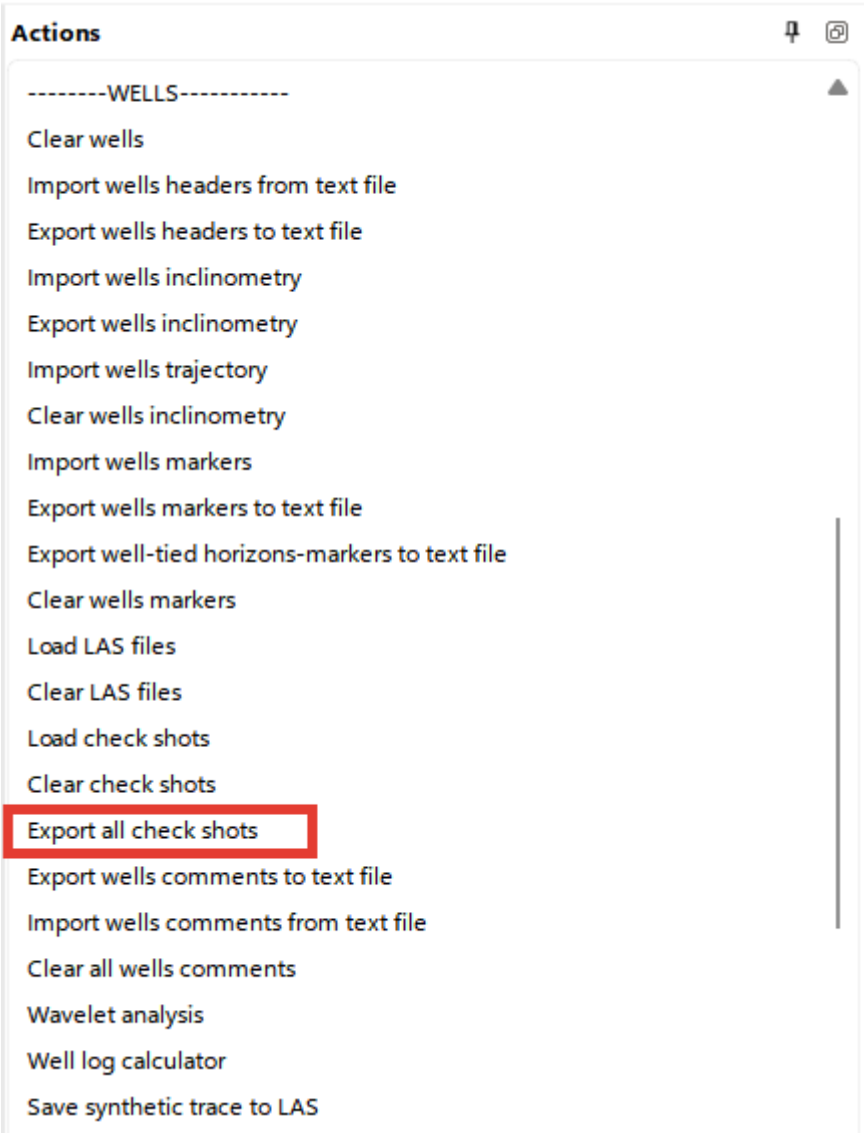
There are two options for exporting checkshots:

1. **Export Checkshots for All Wells**
   - To export checkshots for all wells:
     - In the **Data Manager**, right-click on the **Wells** icon and select **Export** > **Checkshots**.

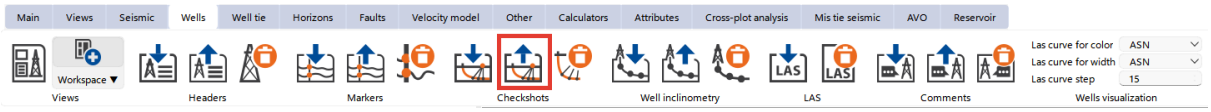


     - Alternatively, go to the **Actions** panel and choose **Export all checkshots**.

© 2026 Geomage 2003 LTD

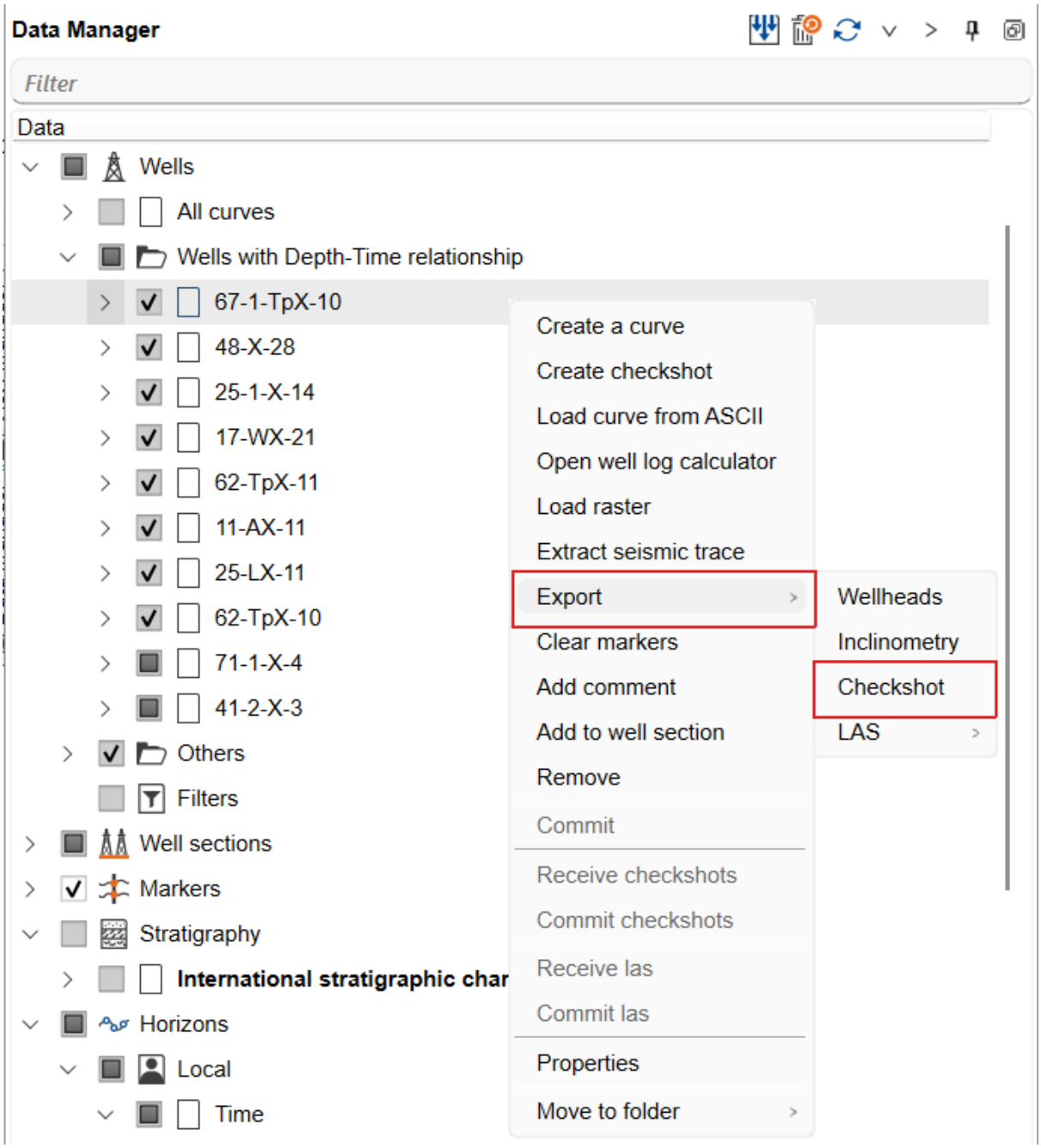


Or go to the Wells Bar and choose **Export all checkshots**.



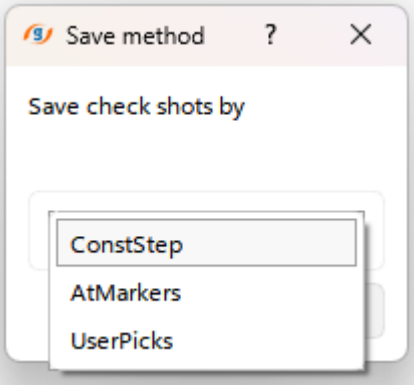
2. **Export Checkshots for a Single Well**
   - To export checkshots for an individual well:
     - In the **Data Manager**, right-click on the specific well name, then select **Export** > **Checkshot**.

© 2026 Geomage 2003 LTD



When exporting checkshots, a pop-up window will appear with options to save checkshots by different methods:

© 2026 Geomage 2003 LTD



- **ConstStep**: Save at constant depth intervals.
- **AtMarkers**: Save checkshots at predefined markers.
- **UserPicks**: Save checkshots at user-defined depths or picks.

Choose the desired method, then locate the directory and file format for saving, and click **Save**.

The exported file will include the following information by default:

- UWI (Unique Well Identifier)
- True Vertical Depth (TVD)
- One Way Time (OWT)

***Note**: Currently g-Space supports export checkshot data from KB datum.*

For more information about checkshot format please refer to chapter Well Data Formats 507

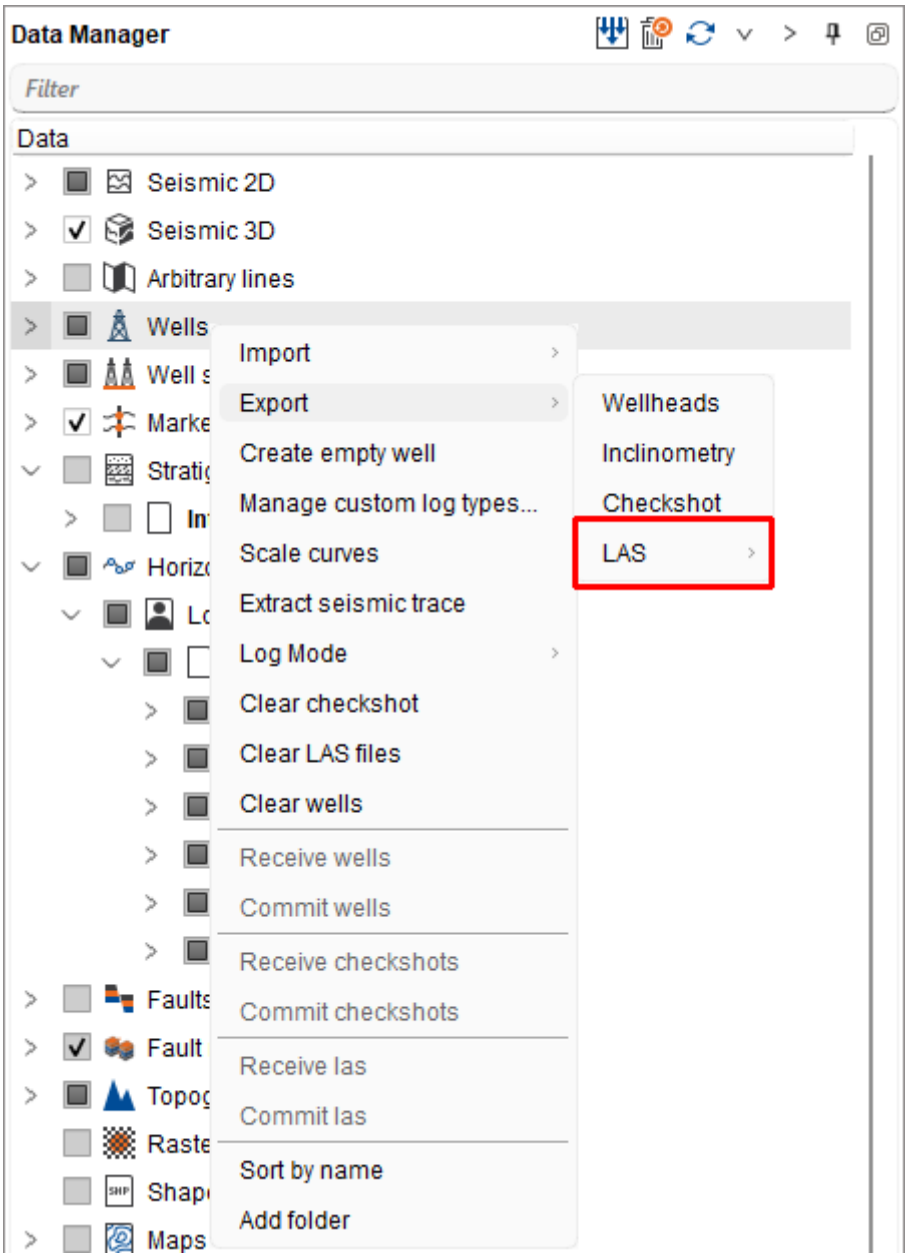
6.6 Export las

There are two options for exporting LAS files: **Export for All Wells** or **Export for a Single Well**.

Export LAS for All Wells

1. In the **Data Manager**, right-click on the **Wells** icon.
2. Select **Export** > **LAS**.

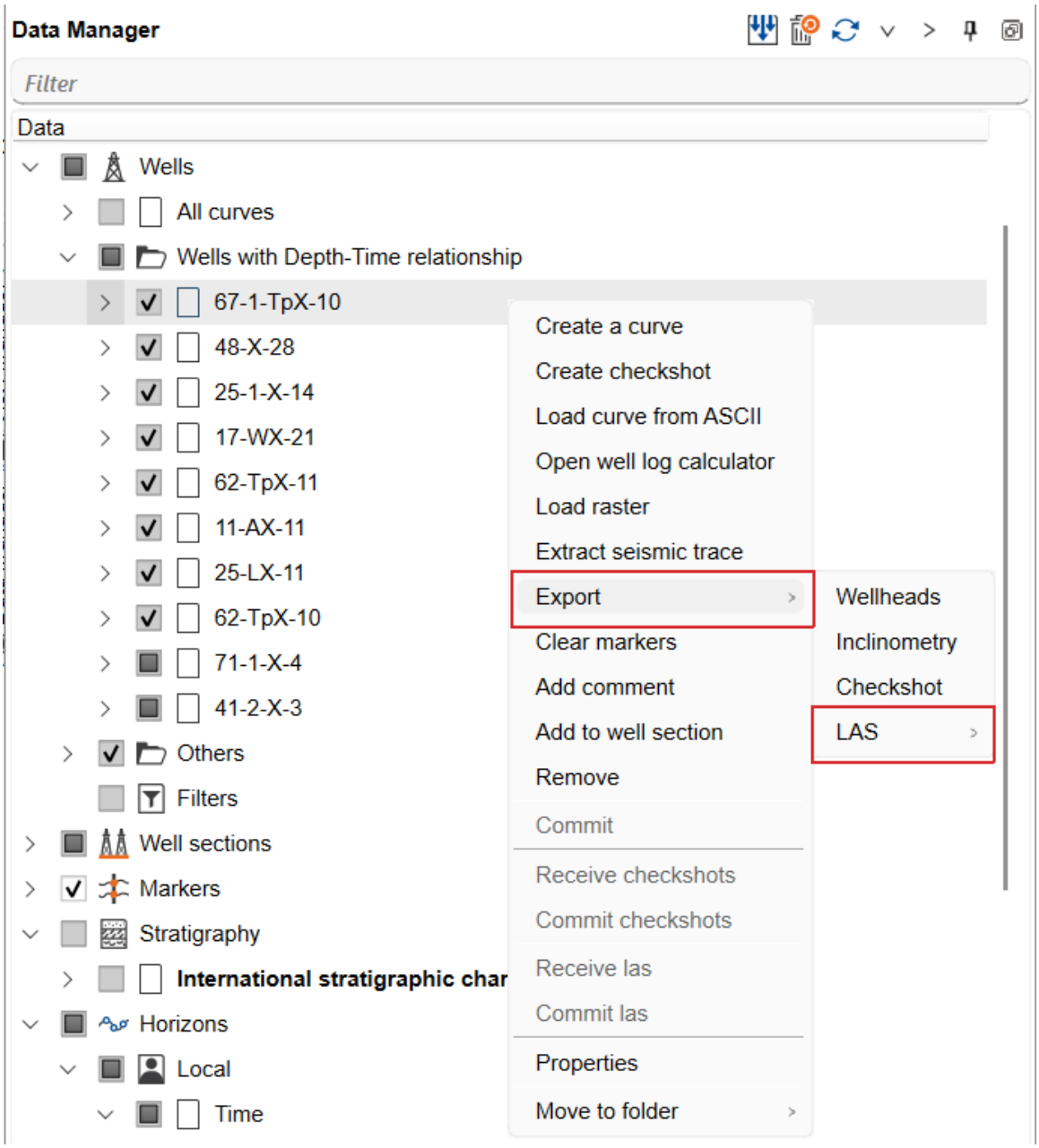
© 2026 Geomage 2003 LTD



Export LAS for a Single Well

1. In the **Data Manager**, locate and right-click on the specific well name.
2. Select **Export** > **LAS**.

© 2026 Geomage 2003 LTD



LAS Export Details

When exporting LAS files, the exported file will be in **Log ASCII Standard (LAS)** format and will include all available curves for each well, providing a complete log dataset for analysis.

The **LAS** export menu offers two output formats:

- **LAS format (*.las)** — standard Log ASCII Standard (LAS 2.0) text file.
- **Excel format (*.xlsx)** — the same log data written as a Microsoft Excel workbook.

© 2026 Geomage 2003 LTD

After choosing a format, g-Space opens a field/curve selection wizard where you select which curves and fields to include in the export before the file is saved.

Depth units (meters or feet)
LAS depth values are written in the units of the project measurement system 365. The **Save curves to LAS file** wizard shows **Min depth**, **Step** and **Max depth** in metres (m) when the project uses the Metric system, and in feet (ft) otherwise. To change the export units, change the project measurement system.

For more information about LAS format please refer to chapter Well Data Formats 507

6.7 Export comment log

Each well in a g-Space project can carry an arbitrary number of comments — free-form notes added by interpreters during the work. The **Export comment log** action writes all well comments to a tab-separated text file so they can be archived, shared, or re-imported on another workstation.

How to export

1. Option 1. From the Action panel 214 select **Export well comments to text file**.
2. Option 2. In the Wells Bar 46 open the **Comments** section and press the export button.
3. Choose a target **.tsv** file in the save dialog. g-Space writes one comment per line, listing the well UWI, the measured depth (MD) and the comment text.

Output format

The exporter produces a tab-separated values (**.tsv**) file suitable for Excel or any text editor. The file begins with the header row **UWI**, **MD**, **Comment**, followed by one row per comment with these three tab-separated columns. For the expected layout see Well Data Formats 507.

See Also

- Well Data Formats 507
- Wells Bar 46

© 2026 Geomage 2003 LTD

- Actions 214

6.8 Export horizons

Note

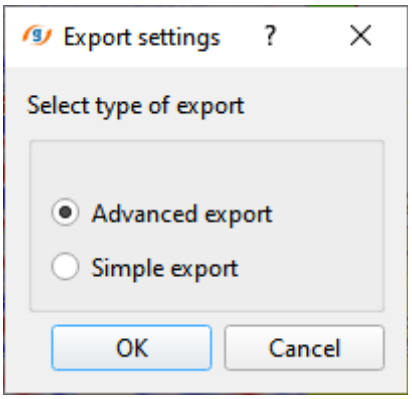
Export horizons will be performed on current datum and replacement velocity. To check these values go to Module Properties 197 panel --> Parameters tab

There are a few options for exporting horizons:

On the Interpretation bar 96, use the **Horizons Management** group for the first two options:

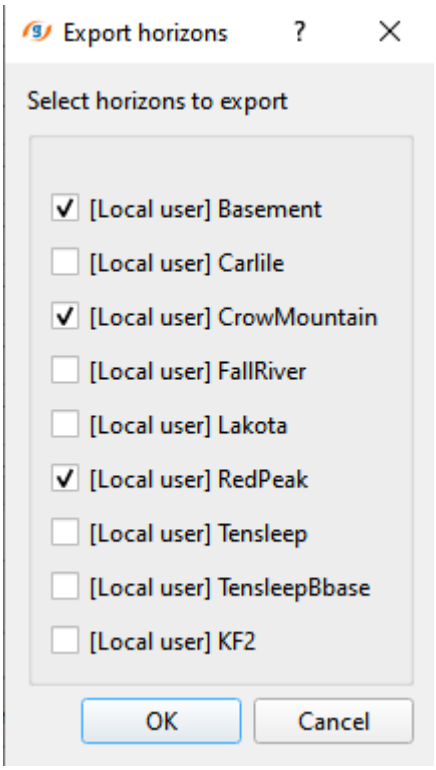
Option 1: Save horizons binary
In case of choosing first option g-Space will clone all horizons set existing in the project.

Option 2: Export current horizon ASCII
Second option supports exporting current horizon (simple export) and set of horizons with controlled parameters (advanced export)

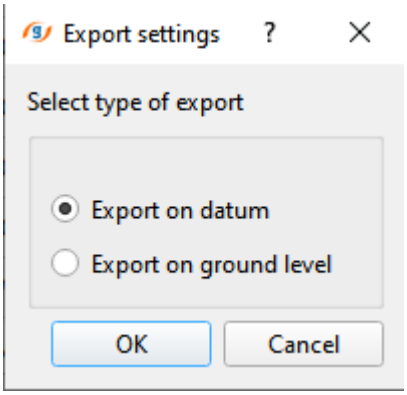


Simple export - will export current horizon with project datum
Advanced horizon supports multi-horizon export

© 2026 Geomage 2003 LTD

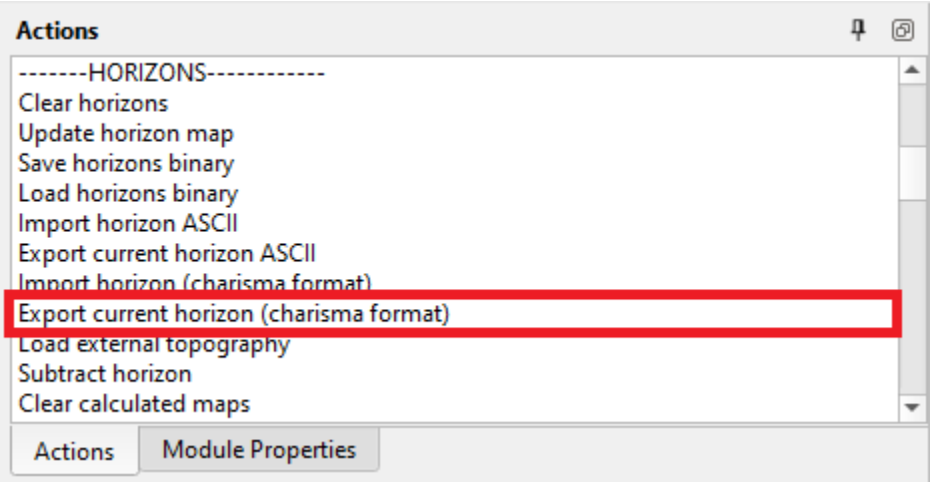


with recalculated datum



Option 3 exports the current horizon in Charisma format. In the Actions panel 214, choose **Export current horizon (charisma format)**.

© 2026 Geomage 2003 LTD



For more information about horizon format please refer to chapter Horizon Data Format 513

Exporting several horizons to Charisma

Several horizons can be written to Charisma format in one step from the Data manager 124. Select the horizons in a **Time** or **Depth** folder, right-click, and choose **Export horizon** > **To Charisma...**
g-Space asks for a **destination folder** and a single **Datum** value for the whole batch, then writes one file per horizon into that folder. Each file is named after its horizon (spaces replaced with underscores) with a `.dat` extension. If any target file already exists, g-Space lists the affected files and asks whether to overwrite them.

6.9 Export faults

Note

Export faults will be performed on current datum and replacement velocity. To check these values go to Module Properties 197 panel --> Parameters tab

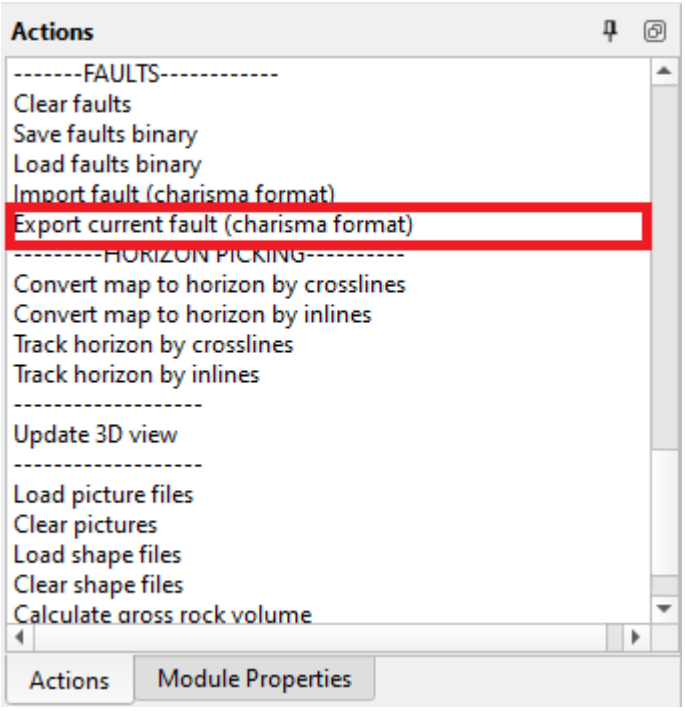
Exporting faults:
On the Interpretation bar 96, use the **Faults Management** group. Two export options are available:

© 2026 Geomage 2003 LTD

Option 1: Press **Save faults binary** button and g-Space will export a copy of all faults

Option 2: Press **Export current fault (charisma format)** button , set the datum and save the file

Another option is to export faults in Charisma format: In Actions panel 214 choose Export current fault (charisma format)



For more information about fault formats supported by g-Space refer to Fault Data Formats 515

Exporting faults from the Data manager

Faults can also be written to Charisma format directly from the Data manager 124. In the **Faults** section, right-click the item you want to export and choose **Export** > **To Charisma...**:

One fault - g-Space asks for the datum in the **Set the required datum** dialog, where **Datum** is prefilled with the current time or depth datum, then asks for a file name and writes one Charisma file.

© 2026 Geomage 2003 LTD

Several selected faults - all faults in the selection are exported in one step.

The Faults folder - every fault in the project is exported in one step.

For a selection of faults and for the whole **Faults** folder, g-Space first asks for a target folder in the **Export faults** dialog and then asks for the datum once for the whole batch. If any target file already exists, g-Space lists those files and asks whether to overwrite them. One file per fault is written into the chosen folder, named after the fault with the extension `.dat`; spaces in a fault name are replaced with underscores.

To export a fault's triangulated surface instead of its sticks, expand the fault in the Data manager 124, right-click its **Surface** node and choose **Export to GOCAD TSurf**. See GOCAD Data Formats 519.

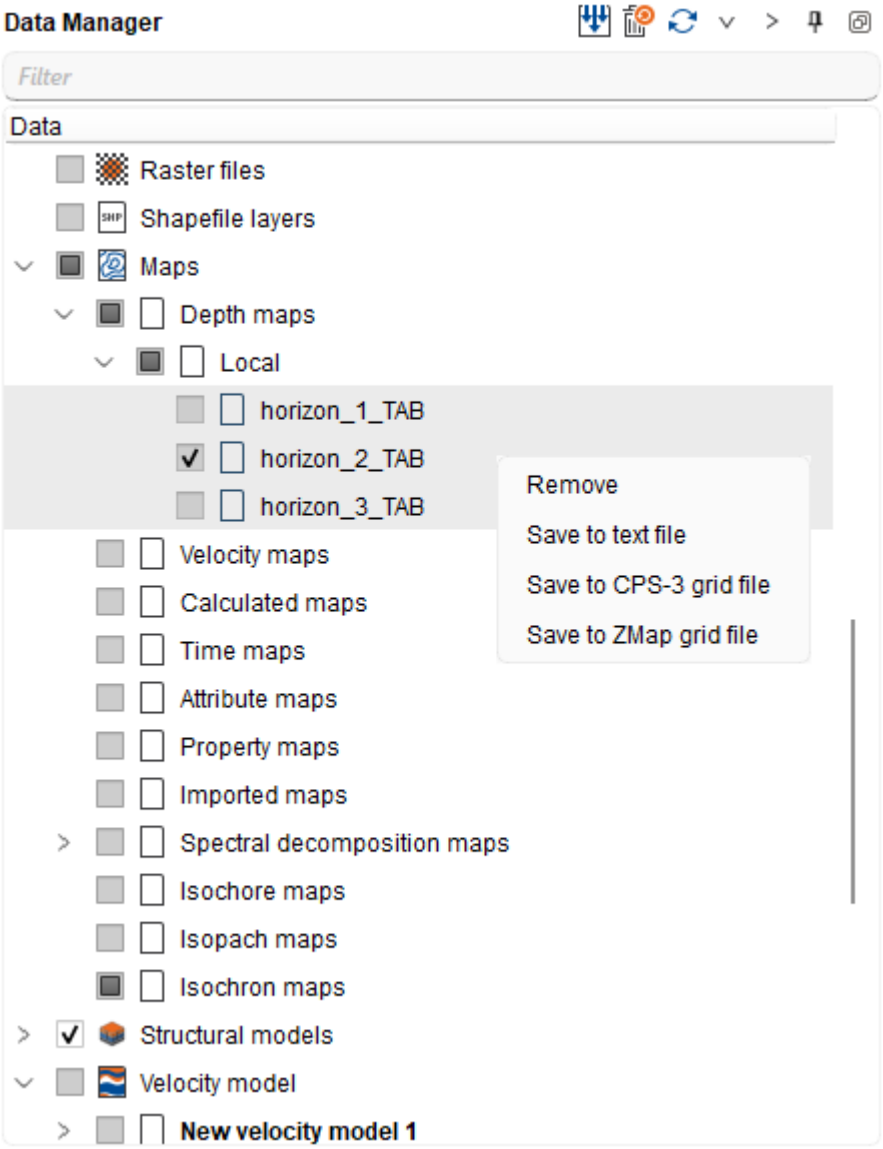
6.10 Export maps

To export **Maps** stored in g-Space go to Maps in Data manager 124 and choose the Map you want to export

Right-click on the Map and choose one of the following options:

1. Save to text file
2. Save to CPS-3 grid file
3. Save to ZMap grid file

© 2026 Geomage 2003 LTD



To export several maps in one step, select them in Data manager 124 (hold Ctrl and left-click each map), then right-click the selection and choose the same export option.
g-Space asks only for the output folder. Every selected map is written to its own file, named after the map name shown in Data manager, with the extension of the chosen format: .txt for text files, .cps3 for CPS-3 grid files and .dat for ZMap grid files.
For text files, g-Space asks once whether to write invalid values and applies your answer to all selected maps. A progress bar shows the export and can be stopped. If some maps cannot be saved, their names and the reasons are listed in a message when the export finishes.

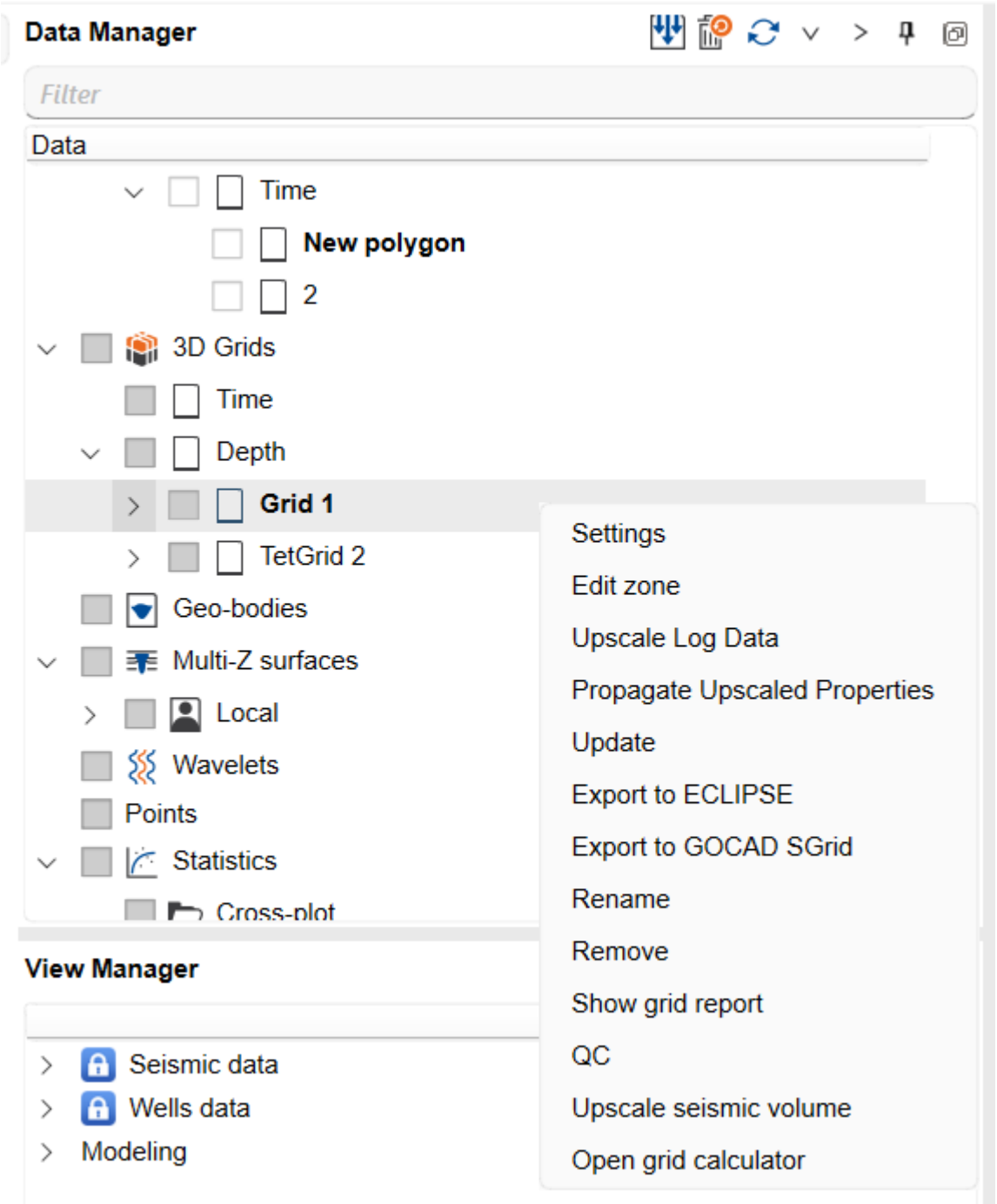
For more information about maps formats supported by g-Space refer to Maps Data Formats 514

© 2026 Geomage 2003 LTD

6.11 Export grids

To export **Grids** stored in g-Space navigate to 3D Grids in Data manager, locate Grid you want to export

Right-click on the Grid, and there is an option to export the grid to Eclipse format



© 2026 Geomage 2003 LTD

The export process will generate:

- A **.dat file** containing general grid information and property data.
- A **.GRDECL file** storing grid coordinates and structure.

The exported file declares its unit system in the header: **FIELD** when the project uses **Imperial** units and **METRIC** otherwise. Grid coordinates and depths are written in the units of the project measurement system, so an exported grid keeps its original position when it is imported back into g-Space.

For more information about maps formats supported by g-Space refer to Grid Data Formats 519

6.11.1 Export 3D models

g-Space exports 3D grids to the **GOCAD** / SKUA-GOCAD ASCII exchange formats. A structured grid is written as a GOCAD **SGrid** (`.sg`) and a tetrahedral grid as a GOCAD **TSolid** (`.so`), so a model built in g-Space can be handed to GOCAD / SKUA and other packages.

Purpose

Write a g-Space grid and its scalar properties to a GOCAD ASCII file for exchange with third-party software. The export preserves node coordinates and per-node or per-cell properties, and records the time / depth domain and the Z sign convention in the file header so the grid round-trips back into g-Space.

Launching the export

In the Data manager 124, right-click the grid you want to export under the **Grids** node. The available GOCAD action depends on the grid type:

- **Export to GOCAD SGrid** — offered for a structured (S-Grid) grid.
- **Export to GOCAD TSolid** — offered for a tetrahedral grid.

Step-by-step

1. In the Data manager 124, right-click the target grid under the **Grids** node.
2. Choose **Export to GOCAD SGrid** for a structured grid, or **Export to GOCAD TSolid** for a tetrahedral grid.

© 2026 Geomage 2003 LTD

3. In the save dialog, give the output file a name and location (`.sg` for an SGrid, `.so` for a TSolid).
4. g-Space writes the GOCAD header with the grid dimensions, domain and property list, plus the geometry and property values. For an SGrid the per-node coordinates and properties are written to a companion ASCII data file alongside the `.sg` header, so keep the two files together.

Notes

- Structured grids export as an SGrid header plus a companion ASCII data file; tetrahedral grids export as a single TSolid file containing the vertices, tetrahedra and per-cell properties.
- The time / depth domain and the Z sign convention (ZPOSITIVE Depth versus Elevation) are written into the header, so an exported file re-imports into g-Space with the same geometry and domain.
- GOCAD Voxet is supported for import only; there is no Voxet export. Export a grid as an SGrid instead.

See Also

Model Data Formats 522 | Import 3D models 453 | GOCAD Data Formats 519

6.11.2 Export RESQML

RESQML is the Energistics reservoir data-exchange standard. g-Space writes polygons, point sets, Multi-Z surfaces and tetrahedral grids to RESQML `.epc` packages that other subsurface applications can read. Each data category is exported separately, into its own package.

Purpose

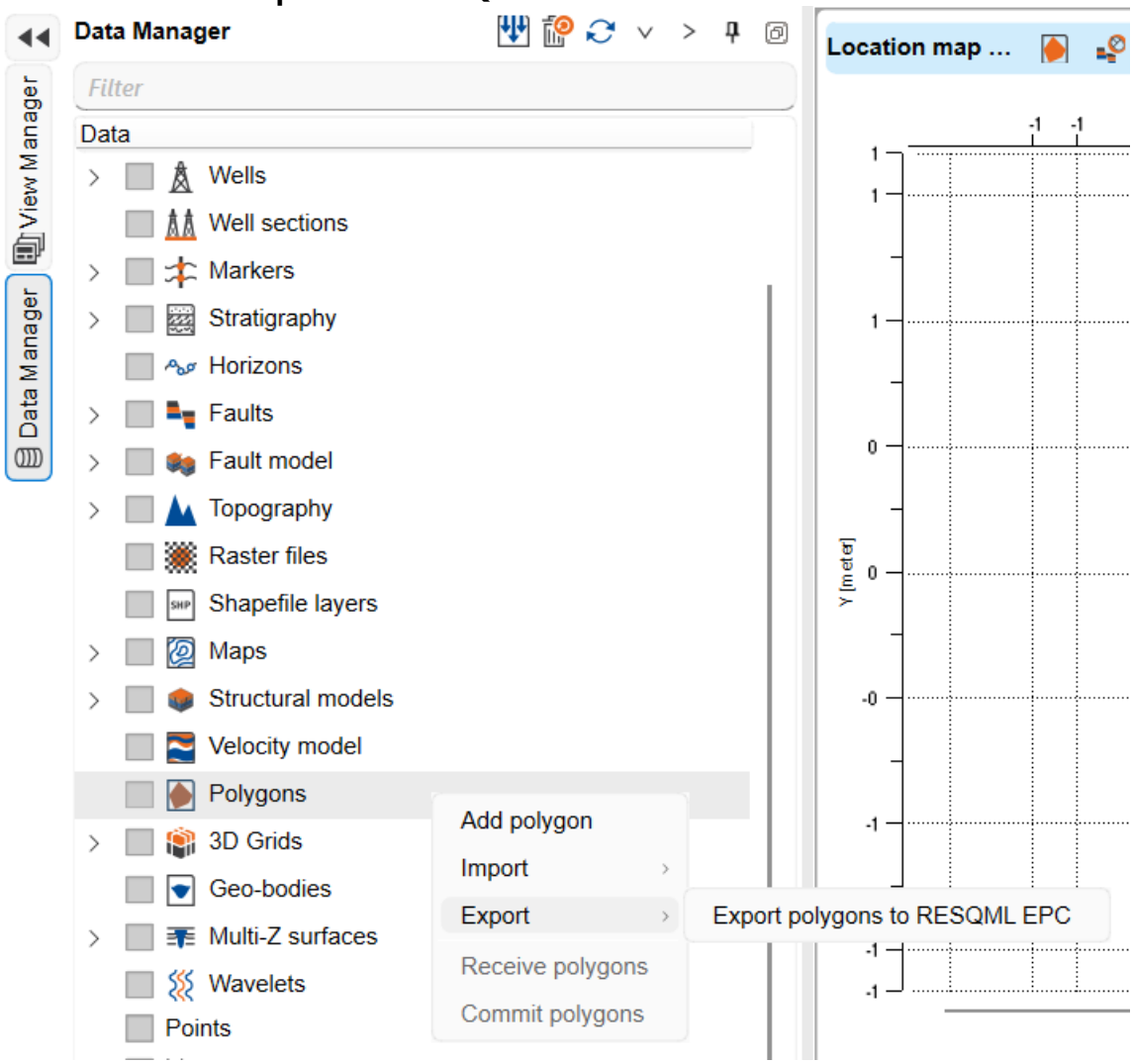
Package one or more g-Space objects into a standard RESQML .epc file so they can be exchanged with reservoir modelling packages that support RESQML.

Launching the export

RESQML export is launched per data category from the Data manager 124 tree. Right-click the branch root that holds the data you want to write and choose its RESQML export action; the command writes every supported object of that branch into one package:

© 2026 Geomage 2003 LTD

- **Polygons** root -> **Export** -> **Export polygons to RESQML EPC**.
- **Points** root -> **Export all to RESQML EPC...**.
- **Multi-Z surfaces** root -> **Export all to RESQML EPC**.
- **3D Grids** root -> **Export all to RESQML EPC**.



Step-by-step

1. In the Data manager, right-click the branch root of the data category you want to export.
2. Choose the RESQML export command listed above for that branch.
3. In the save dialog set the filter to **RESQML EPC (*.epc)**, choose a target `.epc` file and confirm.
4. Review the completion message: it reports how many objects were written, or why the package could not be created. If the branch holds nothing that can be exported, g-Space reports that instead of writing a file.

© 2026 Geomage 2003 LTD

Notes

- Each export writes its own package: one `.epc` file plus a linked `.h5` file of the same name beside it, which holds the bulk arrays. Keep the two files together when you copy or send the result.
- Exporting to a file name that already exists replaces it, so give each data category its own file name.
- From the **3D Grids** branch only tetrahedral grids are written, together with their cell properties. Grids of other types are skipped.
- Geometry and grid properties are written; g-Space display styling is not part of the RESQML package.

See Also

Model Data Formats 522 | Import RESQML 456 | Export to VTK 488

6.11.3 Export to VTK

VTK is a widely supported visualization format. g-Space can export a tetrahedral grid to a `.vtk` file as an unstructured mesh together with its properties, for use in external visualization and analysis tools.

Purpose

Write a tetrahedral grid (and its property arrays) out as a VTK unstructured mesh so it can be viewed or post-processed outside g-Space. VTK is export only.

Launching the export

In the Data manager 124, right-click the tetrahedral grid and choose the **Export to VTK** command, then select the target `.vtk` file.

Step-by-step

1. Select the tetrahedral grid in the Data manager.
2. Choose **Export to VTK** from the context menu.

© 2026 Geomage 2003 LTD

3. Pick the output `.vtk` file and confirm. The mesh and its properties are written as a VTK unstructured grid.

Notes

- Only tetrahedral (unstructured) grids can be exported to VTK.
- To exchange the same grid in a reservoir format, use Export RESQML 486 or ECLIPSE / GRDECL export instead.

See Also

Model Data Formats 522 | Export RESQML 486 | Export grids 484

6.11.4 Export DXF

DXF (Drawing Exchange Format) is the AutoCAD interchange format. g-Space exports polygon and geobody geometry to `.dxf` files so the data can be opened in AutoCAD and other CAD/GIS packages. Writing is handled internally by the bundled `libdxfrw` library.

Purpose

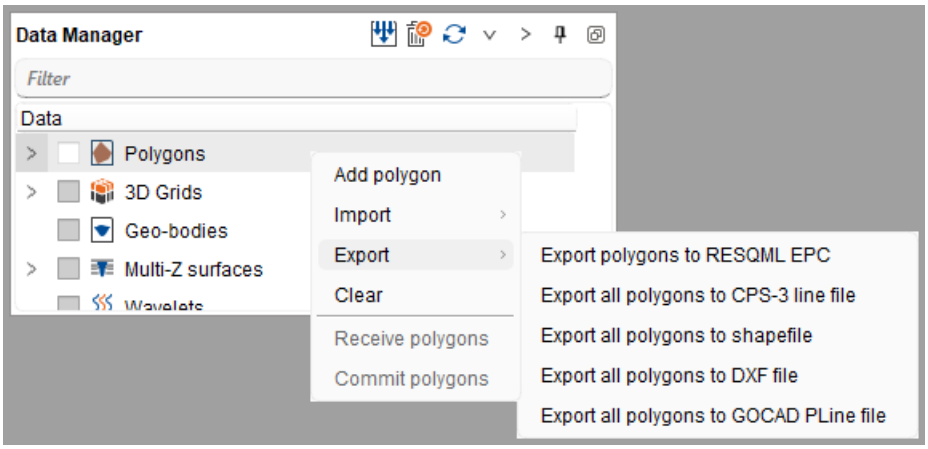
Write g-Space polygons (and geobody outlines) out as AutoCAD DXF geometry. Each polygon is exported as a closed polyline on its own layer, with a text label at the polygon centroid. Export requires a Seismic Interpretation license.

Launching the export

Export is launched from the Data manager 124 context menu. Right-click one or more polygons, or the polygons group, and choose one of:

- **Export to DXF file** - exports the selected polygon.
- **Export all polygons to DXF file** - exports every polygon in the group. On the **Polygons** category this command is in the **Export** submenu.

© 2026 Geomage 2003 LTD



Step-by-step

1. Select the polygon(s) to export in the Data manager 124 and open the context menu.
2. Choose **Export to DXF file** or **Export all polygons to DXF file**.
3. In the **Save Polygons to DXF file** dialog, pick a location and file name. The `.dxf` extension is added automatically if you omit it.
4. Confirm to write the file. Open the resulting `.dxf` in your CAD/GIS application to verify the geometry.

Notes

- Files are written in AutoCAD `AC1024` (2010) DXF format.
- Each polygon becomes a closed `LWPOLYLINE` placed on a layer named after the polygon's owner (or the polygon name when no owner is set).
- A `TEXT` label with the polygon name is written at the polygon centroid on a dedicated `Labels` layer.
- Polygon coordinates are written in the project coordinate system; DXF stores no CRS metadata, so record the units and datum separately if the file will be shared.

See Also

Model Data Formats 522 | Import DXF 458 | Export polygons 491

© 2026 Geomage 2003 LTD

6.12 Export polygons

Polygons created or imported in g-Space can be exported to exchange formats for use in external mapping and GIS applications, or for archival. Export is driven from the Data manager 124.

How to export

1. Open Data manager 124 and expand the **Polygons** category.
2. Right-click the polygon to export.
3. Choose an export format from the context menu and select a target file in the save dialog.

Supported formats

- **CPS-3 line (.cps3)** — native line format; round-trip with Import polygons 444.
- **Shapefile (.shp)** — standard GIS format. Each polygon is written as a feature inside a Polygons layer, so multiple polygons can be exported into a single shapefile. See Export shapefile 500.
- **GOCAD PLine (.pl)** — export a polygon, or all polygons in a folder, to GOCAD. See GOCAD Data Formats 519.
- **DXF (.dxf)** — AutoCAD polygons and geobodies. See Export DXF 489.

See GIS Data Formats 517 for a detailed description of each format.

See Also

- Import polygons 444
- GIS Data Formats 517
- Data manager 124

6.13 Export mistie cross-points

The **mis-tie cross-points table** is a line-intersection misfit table built during 2D seismic mis-tie correction. g-Space can export it to a tab-separated ASCII `.txt` file so the computed misfits, time shifts, and phase rotations can be reviewed, quality-checked, or reused elsewhere.

© 2026 Geomage 2003 LTD

Purpose

Exporting the cross-points table writes out, for each intersection of two 2D lines, the measured time misfit and the time shift and phase rotation applied to each line, plus a checked (QC) flag. The plain-text output is convenient for QC in a spreadsheet, for archiving a correction, or for re-importing the same set of intersection corrections later.

Launching the export

After computing mis-tie misfits and shifts, use the **Export cross-points table** command. A save-file dialog opens filtered to **Text files (*.txt)**; choose a location and file name for the output.

Step-by-step

1. Run the mis-tie analysis so the cross-points table holds the misfits, shifts, and rotations for the line intersections.
2. Invoke **Export cross-points table** and pick the target `.txt` file.
3. g-Space writes a header line and then one tab-separated row per intersection.
4. Open the file in a text editor or spreadsheet for QC, or keep it to re-import the same corrections into the project later.

File format

The export is tab-separated ASCII with a header row followed by one line per intersection. The ten columns are:

- **X**, **Y** - coordinates of the line-intersection point.
- **Checked** - QC flag, written as `Y` or `N`.
- **Misfit** - measured time misfit at the intersection.
- **Line1**, **Line2** - names of the two intersecting 2D lines.
- **Shift1**, **Shift2** - time shifts applied to each line.
- **Rotation1**, **Rotation2** - phase rotations (in degrees) applied to each line.

This is the same layout the import expects, so an exported file can be edited and loaded straight back with Import mistie cross-points 386.

© 2026 Geomage 2003 LTD

See Also

Import mistie cross-points 386 | Seismic Data Formats 511

6.14 Export mute picking

Mute picking export writes the top and bottom mute curves picked on pre-stack gathers to disk as `.corr` files, so the mutes can be archived or re-imported into another project.

Purpose

Save the top and bottom mute curves currently attached to a pre-stack geometry. Each existing mute curve is exported to its own file so it can later be loaded back with Import mute picking.

Launching the export

In the Data manager 124, right-click a pre-stack seismic geometry that has a mute picked and choose **Save mute picking**. The action appears only when a top or bottom mute exists for that geometry, and is also available as **Save mute** from the pre-stack gather context. It requires the geophysical interpretation license.

Step-by-step

1. Right-click the pre-stack geometry in the Data manager 124 and select **Save mute picking**.
2. In the **Save mute picking files to folder** dialog, choose the destination folder. Files are named automatically, so no file name is entered.
3. Press **OK** to write the files. Existing top and bottom mutes are saved as `.corr` files whose names embed the seismic name, the mute type and the geometry hash.

Notes

- The top and bottom mutes are written to separate files, for example `<seismic>_mute_picking_top_<hash>.corr` and `<seismic>_mute_picking_bottom_<hash>.corr`.

© 2026 Geomage 2003 LTD

- Only the mute types that are actually picked for the geometry are exported; a geometry with only a top mute produces a single file.
- The geometry hash in the file name lets Import mute picking 387 match each file back to the correct geometry when several files are loaded together.
- The action is available only with the geophysical interpretation license.

See Also

Import mute picking 387 | Seismic Data Formats 511 | Export data 464

6.15 Export seismic

g-Space writes seismic data back to disk in the **SEG-Y** format (`.sgy` / `.segy`). This topic covers exporting post-stack volumes, 2D lines, arbitrary-line sections and velocity models from a project as standard SEG-Y files.

Purpose

Export seismic amplitude, attribute or velocity data so it can be exchanged with other software or archived. Export is one-directional: the loaded data is written to a new SEG-Y file together with its trace headers and a textual (EBCDIC) header, leaving the project data unchanged.
The following export variants are available, depending on the selected object:

- Post-stack 3D volumes and 2D lines, as loaded.
- Time-to-depth converted volumes and lines (**Export time to depth**).
- Shifted volumes and lines that carry the applied static / bulk shifts (**Export with all shifts**).
- Arbitrary-line sections extracted along a user-drawn path.
- Velocity models.

Launching the export

In the Data Manager 124, right-click the seismic object you want to export (a loaded volume, 2D line, arbitrary line or velocity model) and choose the matching export command from the context menu.
For a plain volume or line the context menu offers commands such as **Export time to depth**, **Export with all shifts** and the batch variants **Export all time lines with shifts** / **Export all time volumes with shifts**. An arbitrary line offers **Export arbitrary line**. Seismic export commands are also reachable from the seismic ribbon while a seismic object is active.

© 2026 Geomage 2003 LTD

Step-by-step

1. In the Data Manager 124, right-click the seismic object and pick the required export command.
2. In the file dialog choose the output folder and file name. The file filter offers `.sgy` and `.segy`.
3. Set the **Trace header formats** template that controls the binary / trace-header byte layout written to the file.
4. Set the writing options: **Saving mode** (`append` or `direct`), **Rewrite file**, **Convert to feet** and **Save text header to separate file** as needed.
5. For a shift or depth export, confirm the datum / conversion parameters; the traces are corrected to the project datum before being written.
6. Run the export. g-Space writes the traces to the SEG-Y file and reports the **Number of traces written** on completion.

Parameters and notes

- Sample format — samples are written as 4-byte floating point, either IBM float or IEEE float, according to the chosen trace-header format.
- Text (EBCDIC) header — a textual header is written with the file; it can optionally be saved to a separate `.txt` / `.csv` / `.ebcdic` file.
- **Scalars** — coordinate, elevation and shot-point scalar values control how header coordinates are encoded.
- **Domain** — both time and depth data can be exported; use **Export time to depth** to write a converted depth volume from a loaded time volume.
- **Shifts** — the **with all shifts** variants bake the applied static and bulk shifts into the exported traces.
- Arbitrary-line export writes the extracted section to a single SEG-Y file after correcting the traces to the output datum.

See Also

Seismic Data Formats 511 | Import seismic 373 | Export seismic to GeoTIFF 496

© 2026 Geomage 2003 LTD

6.16 Export seismic to GeoTIFF

GeoTIFF is a georeferenced raster image format. g-Space can rasterize the inline and crossline sections of a 3D post-stack volume and write them as georeferenced `.tif` images. This is an export-only workflow driven by the **Export 3D data to GeoTiff** wizard.

Purpose

Produce ready-to-map images of selected seismic sections. Each requested inline or crossline is colored with a palette and saved as a separate GeoTIFF that carries the section's real-world coordinates, so the images line up correctly in GIS and mapping software. There is no import back from GeoTIFF.

Launching the export

In the Data Manager, right-click a loaded 3D post-stack volume and choose **Export inlines/crosslines to GeoTIFF**. First select the destination folder, then the **Export 3D data to GeoTiff** wizard opens. Prestack volumes cannot be exported to GeoTIFF.

Step-by-step

1. Right-click the 3D volume and choose **Export inlines/crosslines to GeoTIFF**.
2. Pick the output folder in the **Save GeoTIFF** dialog.
3. In the **Export inlines** group, keep **Export inlines** checked to write inline sections, then set **Start inline**, **Step inline** and **End inline**. The ranges default to the full extent of the volume.
4. In the **Export crosslines** group, do the same for **Start crossline**, **Step crossline** and **End crossline**. Uncheck a group to skip that direction.
5. Choose a **Palette** for the color mapping and a **Normalization** mode (**Seismic** or **Matrix**).
6. Press **OK** to run the export.

Parameters and notes

- **Inline / crossline ranges** — start, step and end select which sections are rasterized; the step lets you export every N-th line.

© 2026 Geomage 2003 LTD

- **Palette** — the chosen color palette maps sample amplitudes to RGB colors in the output image.
- **Normalization** — **Seismic** uses a symmetric range around zero (equal positive and negative limits), while **Matrix** scales to the actual minimum and maximum of the section.
- **Output files** — one 3-band RGB GeoTIFF is written per section, named after the volume with an `IL_` or `XL_` suffix and the line number, e.g. `MyVolume_IL_1200.tif`.
- **Georeferencing** — each image is tagged with ground-control points from the section's bin coordinates and the project base coordinate reference system, so it can be overlaid on a map.
- Sections are corrected to the project depth datum before rasterizing, so all exported images share a consistent datum.

See Also

Image Data Formats 516 | Export seismic 494 | Seismic Data Formats 511

6.17 Export marker ages

Marker ages are the geological age and type assigned to each marker of a marker set. g-Space writes them to a plain, tab-separated text `.txt` file with three columns: Marker, Age and Type. This topic covers exporting that file; the same layout can be edited and read back with Import marker ages 434.

Purpose

Write the current markers of a marker set, together with their ages and types, to a text file for review, editing in a spreadsheet, or transfer to another project. Every marker in the set is exported in its stored order.

Launching the export

In the Data manager 124, right-click the **marker set** to export and choose **Export markers ages**.

Step-by-step

1. Right-click the marker set and select **Export markers ages**.

© 2026 Geomage 2003 LTD

2. In the file dialog, set the filter to **Text file (*.txt)** and choose a name and location.
3. g-Space writes a header line followed by one line per marker, each holding the marker name, its age and its type separated by tabs.

File format

The file begins with a header line `Marker columnAge columnType column` and then one tab-separated line per marker:

- **Marker** - the marker name.
- **Age** - the numeric age value.
- **Type** - integer type code: `0` = Marker, `1` = Fault, `2` = Intrusion.

The layout matches Import marker ages 434, so an exported file can be adjusted and re-imported. Marker ages are part of the geological interpretation license.

See Also

Import marker ages 434 | Export markers 468 | Well Data Formats 507

6.18 Export well logs to Excel

Excel workbooks (`.xlsx`) are a convenient tabular format for reviewing and sharing well log data outside g-Space. This topic covers exporting the log (LAS) curves of one or more wells to Excel; g-Space writes the workbooks but does not read logs back from `.xlsx`.

Purpose

Write the depth-indexed log curves of a well to an Excel workbook. Each exported file contains a header block with the LAS parameters (start, stop, step and null value) and well identification, followed by a curve-name row, a units row and the sampled curve values. Export can run for a single well or for all wells, with a per-curve selection.

Launching the export

In the Data manager 124, right-click a well (or the **Wells** root node to export every well) and open the **Export** submenu, then choose **LAS → Excel format (*.xlsx)**.

© 2026 Geomage 2003 LTD

Step-by-step

1. In the **LAS Export Setup** dialog, tick the header fields to include (**Well Name**, **UWI**, **MD**).
2. Under **Log Curves**, expand each well and tick the curves to export. Use **Select All** or **Deselect All** to toggle everything at once.
3. Press **OK** to confirm the selection.
4. When prompted with *Set directory for output xlsx files*, choose the output folder.
5. g-Space writes one workbook per LAS file per well into that folder, named from the well UWI and the LAS file name.

Workbook layout

Each `.xlsx` file is written on a single sheet with, from top to bottom:

- A header block of well parameters: `STRT`, `STOP`, `STEP`, `NULL`, `COMP`, `WELL`, `UWI` and `APIN` with values and descriptions.
- A blank separator row, then a row of curve names and a row of curve units.
- The sampled curve values, one column per curve.

Notes

- Only the checked curves are written; the depth/index curve is always kept as the first column.
- Exporting from the Wells root processes all wells; exporting from a single well processes just that well. Each LAS file of a well produces its own workbook.
- Excel export requires a Seismic Interpretation or Geological Interpretation license; otherwise the action reports that the feature is not available.
- Export is one-way: g-Space does not import well logs from Excel.

See Also

Export LAS 474 | Well Data Formats 507

6.19 Export points

Point sets and cultural points can be exported from g-Space to plain delimited text (CSV / TAB / `.txt`) for use in external mapping and GIS applications, or for archival.

© 2026 Geomage 2003 LTD

Purpose

Write a point set out as a text table with X and Y coordinates followed by the point attributes, in a layout that can be re-imported or read by other software.

Launching the export

In the Data manager 124, right-click the point set (under the **Points** node) and choose the export command, then pick the output text file.

Step-by-step

1. Right-click the point set in the **Points** node and choose the export-to-text command.
2. Choose the output file and delimiter (CSV / TAB / TXT).
3. Confirm to write the X, Y and attribute columns to the file.

See Also

Point Data Formats 523 | Import points 460 | Export shapefile 500

6.20 Export shapefile

Esri Shapefile is the standard GIS exchange format. g-Space exports shape layers and polygons to a Shapefile (`.shp`) so they can be used in GIS and mapping applications.

Purpose

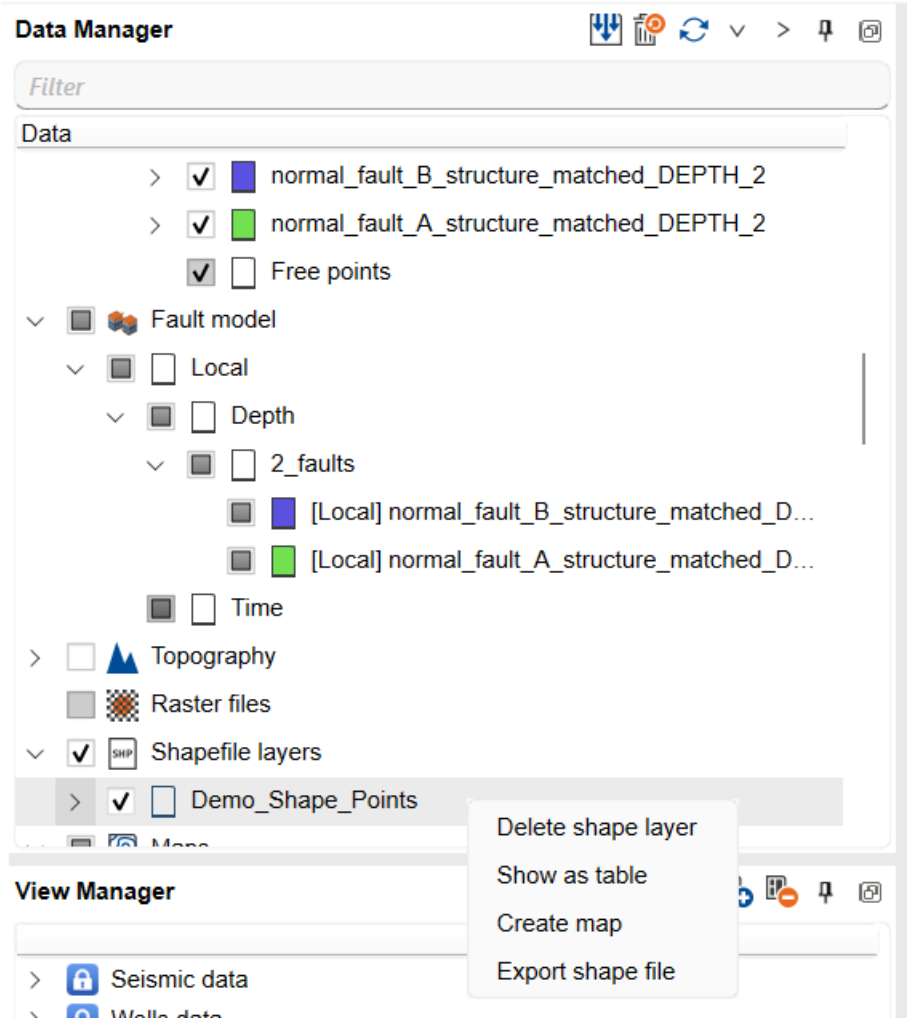
Write polygons or shape layers out as an Esri Shapefile. Multiple polygons are written as features inside a single Polygons layer, so a whole folder can be exported into one shapefile.

Launching the export

In the Data manager 124, right-click a polygon or the **Polygons** folder and choose the Shapefile export command, then pick the target `.shp` file.

© 2026 Geomage 2003 LTD

A loaded shape layer is exported in the same way: expand **Shapefile layers**, right-click the layer and choose **Export shape file**, then pick the target `.shp` file.



Step-by-step

1. Right-click a shape layer under **Shapefile layers**, or a polygon (or the **Polygons** folder to export them all), in the Data manager.
2. Choose **Export shape file** for a shape layer, or the export-to-Shapefile command for polygons.
3. Select the output `.shp` file. The companion .shx and .dbf files are written alongside it.
4. The file is written in the project coordinate reference system.

Notes

© 2026 Geomage 2003 LTD

- Only geometry (and basic attributes) are written; g-Space display styling is not part of the shapefile.
- Shapefiles are imported with Import Shapefile 443.

See Also

GIS Data Formats 517 | Import Shapefile 443 | Export polygons 491

6.21 Export view image

Export view image saves the currently displayed 2D, 3D or well-log view as a raster picture - effectively a high-quality screenshot of the view. g-Space can write `.png`, `.jpeg`, `.bmp`, `.tif` / `.tiff` and animated `.gif` files. This topic covers exporting a view; it is an export-only workflow.

Purpose

Produce a print-ready or presentation-ready image of exactly what is drawn in a view, at a chosen scale, size and resolution. The output is a raster image file only - no interpretation data is written.

Launching the export

With the view you want to capture active, open the **Export view to file** dialog. It shows an export settings panel on the left and a live preview of the result on the right.

Step-by-step

1. Set the output file. Type a path into **File name** or press the browse button (**...**) and choose the image format (PNG, JPEG, BMP, TIF, TIFF or GIF) in the file dialog.
2. Choose the **Export area**: **Entire view**, **Visible** (only what is currently on screen) or **Custom** (enter the horizontal and vertical from / to range).
3. Set the **Horizontal scale** and **Vertical scale** for the exported picture; use **Refresh scales** to reset them to the view's current scale.
4. Under **Image settings**, set the **DPI** (output resolution) and, for JPEG, the **Image quality** (0 = small compressed file, 100 = large near-lossless file).

© 2026 Geomage 2003 LTD

5. Adjust **View settings** as needed: **Print view name**, **Print axes titles** and **White borders**. The **Image info** group reports the resulting pixel and millimetre size.
6. Optionally tick **Enable preview** and press **Preview** / **Fit to window** to check the result before saving.
7. Press **OK** to render and write the image file.

Parameters

- Scale units - the preview rulers can be shown **In pixels**, **In mm**, **In cm** or **In inches**.
- **Animation settings (GIF format)** - only active when the GIF format is chosen; set the **Animation speed (ms)**, the delay between animation frames.
- **Image quality** applies to the JPEG format only; DPI and scale apply to all raster formats.
- **Make default** stores the current settings as the default for future exports; **Restore default** returns to the saved defaults.

See Also

Image Data Formats 516 | Export data 464

6.22 Export to g-Platform

g-Platform is a companion database application that stores project data centrally. This topic covers exporting horizons and polygons from the current g-Space project to a connected g-Platform database, where they are stored as serialized `g-Item` objects.

Purpose

Publish interpretation results — horizons and polygons — from g-Space to a shared g-Platform database so they are available to other users and applications. Each exported object is written into a chosen storage and project as a serialized `g-Item`.

Launching the export

Export requires an active connection to a g-Platform database. Once connected, the **Export data to g-Platform** dialog opens. On the left, the **Storages and Projects** panel lets you choose the destination: pick an external storage from the storage list (the database default is marked **(default)**) and a target project from the project tree.

© 2026 Geomage 2003 LTD

Step-by-step

1. In the **Storages and Projects** panel, choose the destination storage from the combo box and select the target project in the tree.
2. In the **Data list** section, browse the **Available objects** list of horizons and polygons. Use the **Search objects** box to filter by name.
3. Move the objects you want to export into the **Objects to export** list with the **>** button (or **>>** for all). Double-clicking an object moves it too; the **<** and **<<** buttons move objects back.
4. Objects that already exist in the database are shown greyed out. Clear **Show existing objects** to hide them from the list.
5. Press **OK** to export the selected objects to the chosen storage and project, or **Cancel** to abort.

Notes

- Only horizons and polygons are exported; each is serialized as a `g-Item` in the database.
- Objects already present in the target database are marked as grey shadow items, so you can see what has been exported previously without overwriting it unintentionally.
- The storage list is read from the g-Platform database; the default storage is highlighted and pre-selected.

See Also

Import from g-Platform 461 | Export horizons 478

© 2026 Geomage 2003 LTD

Formats

7 Formats

This chapter describes the data formats currently supported by g-Space for import and export, organized by data type.

Seismic Data Formats 511
- SEG-Y / SEGY - 2D and 3D, post-stack and pre-stack (import and export)
- ZGY, GSD, SEG-D, SEG-2, SEG-B, MiniSEED, Seismic Unix, ASCII to seismic (import)
- GeoTIFF and CPS-3 line (export)

Well Data Formats 507
- Well headers, deviation / trajectory, markers, checkshots, comment log
- LAS (continuous, discrete), DLIS, log text tables; export to LAS / Excel
- WITSML, PRODML, OSDU, IHS 297, Petrel headers, well raster log images

Horizon Data Formats 513
- ASCII, binary archive, Charisma; GOCAD TSurf surfaces

Fault Data Formats 515
- Charisma sticks, free sticks (ASCII), binary archive (*.fsurface); GOCAD TSurf

Maps Data Formats 514
- XYZ / ASCII, CPS-3, ZMap / ZMap+, IRAP classic grid

Grid Data Formats 519
- ECLIPSE / GRDECL corner-point reservoir grids

Model Data Formats 522
- GOCAD SGrid / TSolid / Voxet, RESQML EPC, VTK, DXF

GOCAD Data Formats 519
- GOCAD TSurf (*.ts) surfaces, GOCAD PLine (*.pl) polygons

GIS Data Formats 517
- Esri Shapefile, cultural point data, CPS-3 line

Point Data Formats 523
- Point-set / cultural ASCII (CSV / TAB / TXT), GOCAD PLine

Image Data Formats 516
- Geo-referenced raster (world files), satellite imagery; GeoTIFF and view-image export

© 2026 Geomage 2003 LTD

7.1 Well Data Formats

This chapter summarizes well data formats currently supported by g-Space

All table data can be easily copied using **Ctrl+C** and pasted into any external application (e.g., Excel, Notepad) in **table format**.

Well headers format

Well header data can include the well name, Unique Well Identifier (UWI), X and Y coordinates, Kelly bushing (KB), total depth (TD), **Type** and **Status** as separate fields, ground level (GL), company name and completion date.

Supported formats are:
- TAB delimited
- Space delimited
- Comma separated (CSV)
- Petrel well head text files

For more information about loading these formats into g-Space see Import well headers 393

Well deviation (inclinometry) data format

Well deviation contains the information about well trajectory of the well such as Measured Depth (MD), Inclination (INCL), Grid azimuth (AZIM).

	MD	Inclination	Azimuth
1	0	0.0	0
2	1670	2.0	209
3	1741	1.8	206
4	1758	0.0	0
5	3108	2.5	252

Supported formats are:
- TAB delimited
- Space delimited
- Comma separated (CSV)

© 2026 Geomage 2003 LTD

For more information about loading well trajectory into g-Space see Import inclinometry 397

Well markers format

g-Space supports two marker table layouts:

MD layout: provide **UWI** or **Well Name**, **Marker** and **MD**.

	UWI	Marker	MD
1	49025220660000	SHNNu	902.35
2	49025220660000	SSXbase	541.44
3	49025220660000	SSXS	488.36
4	49025220660000	SHNNI	1005.26
5	49025220660000	SHNNbs	1070.76

XY layout: provide **UWI** or **Well Name**, **Marker**, **X**, **Y**, and either **TVD** or **TVDSS**.

Age and **Type** can also be mapped when present.

Petrel well-tops text files are also supported. When their columns are declared between **BEGIN HEADER** and **END HEADER**, g-Space detects the layout and maps the standard columns automatically.

Supported formats are:
- TAB delimited
- Space delimited
- Comma separated (CSV)
- Petrel well-tops text

For detailed field mapping and loading steps, see Import markers 403.

Well markers structure format
This is the g-Space binary format with extension *.msstruct which stores markers hierarchy

For more information about loading markers structure into g-Space see Import markers structure 406

© 2026 Geomage 2003 LTD

LAS format

LAS format contains the information of the well log data (continuous or discrete)
The description of this file format is given here:
https://www.asprs.org/a/society/committees/standards/asprs_las_format_v12.pdf

For more information about loading LAS files into g-Space see Import LAS files 407

Checkshot format

Checkshot data files store depth-time relationships. For a standard table, map **TVD** and **Time**, and optionally **UWI**; the import settings let you identify OWT or TWT and seconds or milliseconds. Petrel checkshot files are detected automatically.

Supported formats are:
- TAB-delimited checkshot tables
- Petrel checkshot files (*.txt, *.dat, *.las, *.checkshot)

For more information about loading checkshots into g-Space see Import checkshots 414

Comment log format

Comments log format contain the information about comments at specific depth in text format. **WELL NAME**-->**MD**-->**Comments (text)**

WELL	MD	Comment
AX-36	345	shale
AX-36	900	lime
AX-36	1745	sand
AB-43	520	lime
AB-43	670	dolo
AB-43	800	anhy

Supported formats are:
- TAB delimited

For more information about loading comment log into g-Space see Import comment log 421

Log Files (Text) format

© 2026 Geomage 2003 LTD

Columnar log data can also be loaded from plain text files using the Log Loading Wizard. Each column holds one log curve sampled against depth.

Supported formats are:
- TAB delimited
- Space delimited
- Comma separated (CSV)

DLIS format
DLIS (*.dlis) files store well log curves in a binary industry format. g-Space reads DLIS with a native reader and loads the curves through the same file dialog used for LAS files.

For more information see Import DLIS 426

WITSML and PRODML formats
WITSML is an XML-based industry standard for well data. On import it can carry wells / wellbores, well headers, logs / curves, deviation and trajectory, and formation markers. PRODML import loads well comments into existing wells and point sets from orphan coordinates.

For more information see Import WITSML 427 and Import PRODML 430

OSDU data model
g-Space can import from the OSDU data platform (JSON *.json). A single import can carry wells, wellbores, deviation / trajectory, logs / curves, markers / tops and point sets.

For more information see Import from OSDU 432

Well trajectory format
In addition to the MD/INCL/AZIM deviation format above, well trajectory stations can be loaded from a *.wt file, and Petrel deviation surveys from *.dev files.

For more information see Import inclinometry 397

Well raster log images

© 2026 Geomage 2003 LTD

Scanned paper logs can be loaded as georeferenced raster images (PNG / JPG / JPEG / BMP / XPM). A single image can be loaded on its own, or many images can be loaded in bulk using an XML manifest.

For more information see Import well log images 436

Export formats

Well data can be exported from g-Space in the following formats:

- **LAS** - well log curves (all wells or selected curves, with a re-sampling option). See Export las 474.
- **Excel workbook** (*.xlsx) - well log curves. See Export well logs to Excel 498.
- **Well headers**, **inclinometry**, **markers**, **checkshots** and **comment log** - as TAB / CSV text tables (see the corresponding Export topics).

7.2 Seismic Data Formats

SEG-Y is the central seismic format in g-Space: it is used for both import and export, and is reused for velocity models and arbitrary-line export. In addition, g-Space can import several other seismic formats, most of which are converted to SEG-Y on load. SEG-Y files store seismic traces along with essential metadata in trace headers. These headers contain crucial information such as inline and crossline numbers, elevation data, coordinates, and other attributes needed for seismic processing and interpretation.

Import formats

- **SEG-Y / SEGY** (*.sgy, *.segy) - 2D lines and 3D volumes, post-stack and pre-stack gathers; time and depth domains.
- **ZGY** (*.zgy) - 3D volumes, 2D lines.
- **GSD** - Geomage Seismic Data (*.gsd) - 2D lines, 3D volumes.
- **SEG-D**, **SEG-2**, **SEG-B** (SN338 / SN339) - pre-stack field/shot gathers, converted to SEG-Y on import.
- **MiniSEED** and **Seismic Unix (SU)** - traces / gathers.
- **ASCII to seismic** (*.txt) - builds 2D lines / 3D volumes from ASCII X/Y/Z/Value points.

Export formats

- **SEG-Y** - post-stack volumes and lines, time-to-depth converted, shifted volumes, arbitrary-line sections, velocity models.
- **GeoTIFF** - 3D post-stack inline/crossline sections as georeferenced raster images.
- **CPS-3 line file** - seismic line geometry (inline/crossline positions).

© 2026 Geomage 2003 LTD

Note

Post-stack and pre-stack are auto-detected on SEG-Y import. Pre-stack gather import is license-gated (GEOPHYSICAL_INTERPRETATION).

Notes

SEG-Y files are binary formatted and structured in a way that data is stored in fixed byte locations within trace headers. The trace header is a 240-byte section preceding each seismic trace that holds metadata about that specific trace. Different software applications may use different byte positions for storing similar types of data, leading to variations in SEG-Y file formats. To address this, g-Space provides multiple preinstalled formats and allows users to define custom formats when necessary.

Preinstalled SEG-Y Formats in g-Space

To accommodate variations in SEG-Y files, g-Space includes several predefined formats. Each format specifies particular byte positions for critical data fields. The available formats are:

- **g-Space format** – Default format optimized for g-Space workflows.
- **Geomage format** – Used for data processed within Geomage software.
- **SegFormat** – A generic SEG-Y structure used in various industry applications.
- **Seg-Y, rev.1, 2002** – Standardized SEG-Y format based on the 2002 revision.
- **Geomage Office Post** – A format designed for post-processing seismic data.
- **Coordinates Only** – Contains only coordinate-related information in the trace headers.
- **Constant Elevation** – Used when elevation values are uniform across the dataset.
- **teapot_dome_3d** – A predefined format for specific datasets.
- **Kingdom format** – A format adapted for compatibility with Kingdom seismic interpretation software.

Managing Formats

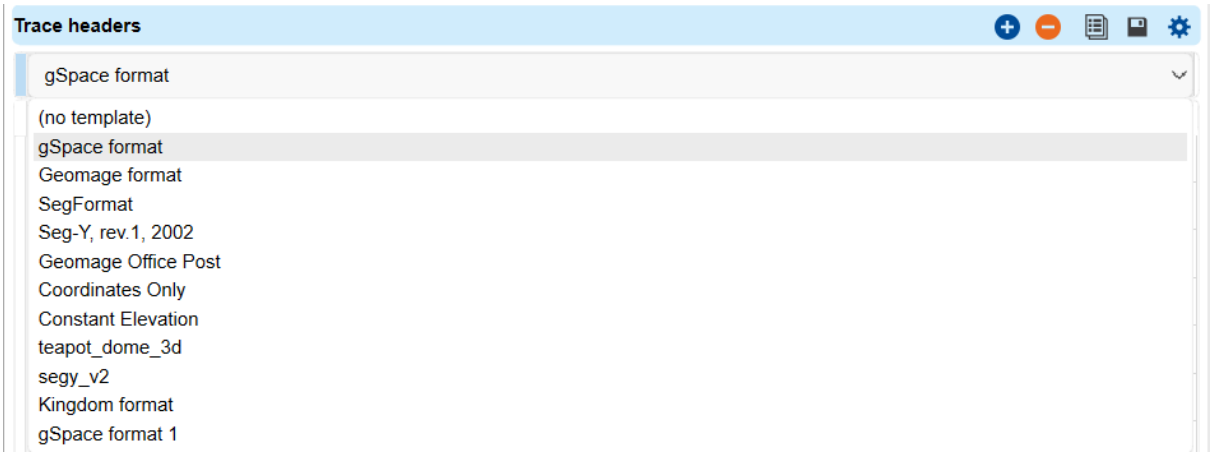
Users can manage and customize SEG-Y formats in the Module Properties 197 under **SEG-Y format settings** and **Trace Header Format settings**. If the required format is not available among the preinstalled options, users can create a new format by specifying the correct byte positions for the necessary trace header information.

Importing Seismic Data

When importing seismic data, g-Space allows customization of geometry handling. Users should ensure that all relevant trace header information is properly configured before loading the data to facilitate seamless interpretation and analysis.

© 2026 Geomage 2003 LTD

The data should have all geometry in trace headers and such information can be customized during the loading process of Import seismic 373



7.3 Horizon Data Formats

Horizon binary data format

It is the internal g-Space format which can be used to shear information between workstations. It stores exact copy of all objects

---

Charisma horizon interpretation format

In header Horizon name.
The data has tab delimited format: Line name --> Point ID (if applicable) --> X --> Y --> TWT in milliseconds --> Seismic folder/group

```
#k2t
bp-2185__poststm__0103__0001.gsd    -100.00    182008.04000    583745.74000    -152.31716    PoSTM_std_NEW
bp-2185__poststm__0103__0001.gsd    -100.00    182008.04000    583745.74000    -152.31716    PoSTM_std_NEW
bp-2185__poststm__0103__0001.gsd    -100.00    182008.04000    583745.74000    -152.31716    PoSTM_std_NEW
bp-2185__poststm__0103__0001.gsd    -100.00    182146.43000    584039.62000    -161.11685    PoSTM_std_NEW
bp-2185__poststm__0103__0001.gsd    -100.00    182146.43000    584039.62000    -161.11685    PoSTM_std_NEW
bp-2185__poststm__0103__0001.gsd    -100.00    182146.43000    584039.62000    -161.11685    PoSTM_std_NEW
bp-2185__poststm__0103__0001.gsd    -100.00    182146.43000    584039.62000    -161.11685    PoSTM_std_NEW
bp-2185__poststm__0103__0001.gsd    -100.00    182175.55000    584108.75000    -147.91731    PoSTM_std_NEW
bp-2185__poststm__0103__0001.gsd    -100.00    182175.55000    584108.75000    -147.91731    PoSTM_std_NEW
```

Horizon interpretation point format (ASCII)

In file header of this format contain Horizon name, information about Replacement velocity and datum. g-Space imports and exports in project units.
The data has tab delimited format: X --> Y --> TWT in milliseconds

© 2026 Geomage 2003 LTD

```
#[Local user] Horizon
#Replacement velocity 2000.000000 m/s
#Points on datum 1000.000000
185048.510000   578265.730000   1644.780138
184974.950000   578225.170000   1648.231571
184827.860000   578144.010000   1665.488733
184754.320000   578103.380000   1675.843030
184625.670000   578032.310000   1686.197328
184497.010000   577961.160000   1689.648760
184441.890000   577930.680000   1694.825909
184331.580000   577869.800000   1700.003057
184276.450000   577839.330000   1698.277341
184147.780000   577768.210000   1694.825909
185103.680000   578296.150000   1643.054422
```

Related surface formats
Horizons and surfaces are also exchanged as triangulated GOCAD TSurf (*.ts) surfaces and as XYZ / grid text. See GOCAD Data Formats 519 and Maps Data Formats 514.

7.4 Maps Data Formats

Grid point data and **external topography format**
These formats are tab delimited and contains X --> Y --> Z.

X	Y	Z
106141.4	417319.1	494.876
106140.6	417232.3	492.402
106139.7	417145.4	490.259
106138.8	417058.6	491.842

Note

Parameter Z may have depth, time, velocity domains. It depends in which folder it is loaded.

For more information about importing this format into g-Space refer to: Import topography 445, Import grids 450

CPS-3 grid data format
Here is the file example

© 2026 Geomage 2003 LTD

```
FSASCI 0 1 COMPUTED 0 -9999
FSATTR 0 0
FSLIMI 69249.028989 270249.028989 455368.532517 673368.532517 0.000000 2612.341309
FSNROW 437 403
FSXINC 500.000000 500.000000
->[Local user] Judea_SV_hor_mark_cs
-9999.000000    -9999.000000    -9999.000000    -9999.000000    -9999.000000
-9999.000000    -9999.000000    -9999.000000    -9999.000000    -9999.000000
-9999.000000    -9999.000000    -9999.000000    -9999.000000    -9999.000000
-9999.000000    -9999.000000    -9999.000000    -9999.000000    -9999.000000
-9999.000000    -9999.000000    -9999.000000    -9999.000000    -9999.000000
-9999.000000    -9999.000000    -9999.000000    -9999.000000    -9999.000000
```

For more information about importing this format into g-Space refer to Import grids 450

ZMap / ZMap+ grid format

A regular ASCII grid in the Petrel / IHS ZMap+ format (*.dat). Import and export are both supported.

For more information refer to Import grids 450 and Export maps 482

IRAP classic grid format

The Roxar / RMS IRAP classic ASCII grid (*.irap / *.gri / *.txt), identified by a -996 magic number in the header. Import only.

For more information refer to Import grids 450

Note

3D reservoir grids (ECLIPSE / GRDECL) are described in Grid Data Formats 519, and 3D model / geobody formats in Model Data Formats 522.

7.5 Fault Data Formats

Fault binary data format

It is the internal g-Space format (*.fsurface) which can be used to share information between workstations. It stores an exact copy of all fault objects (all faults / fault surfaces).

Fault sticks charisma format

File example

© 2026 Geomage 2003 LTD

```
INLINE-    61    572    148538.23000    583150.77000    722.90891    gerar_main    ds-0514_220
INLINE-    61    565    148785.90000    582903.48000    926.11104    gerar_main    ds-0514_220
INLINE-    61    557    149067.35000    582619.24000    1173.28441    gerar_main    ds-0514_220
INLINE-    61    549    149347.75000    582333.98000    1375.83423    gerar_main    ds-0514_220
INLINE-    61    544    149522.47000    582155.18000    1623.65997    gerar_main    ds-0514_220
INLINE-    61    540    149661.97000    582011.85000    1845.27326    gerar_main    ds-0514_220
INLINE-    63    169    168448.31000    587516.51000    582.41302    gerar_main    ds-0516_01
INLINE-    63    171    168504.19000    587433.58000    765.97565    gerar_main    ds-0516_01
INLINE-    63    177    168671.78000    587184.75000    1872.82383    gerar_main    ds-0516_01
INLINE-    64    309    158665.96000    586302.26000    354.43261    gerar_main    ds-0517_269
INLINE-    64    300    158924.28000    585933.79000    714.34766    gerar_main    ds-0517_269
INLINE-    64    293    159126.57000    585648.16000    905.68447    gerar_main    ds-0517_269
INLINE-    64    281    159473.47000    585158.62000    1479.69484    gerar_main    ds-0517_269
INLINE-    64    262    160015.98000    584378.76000    2093.09816    gerar_main    ds-0517_269
```

For more information about importing this format refer to Import faults 441

Fault free sticks (ASCII)

Fault free sticks can be imported from a plain text file (*.txt) through the text-table import wizard, in time or depth domain.

GOCAD TSurf fault surfaces

Triangulated fault surfaces are exchanged with GOCAD / SKUA-GOCAD as *.ts files. See GOCAD Data Formats 519.

7.6 Image Data Formats

Geo-referenced image format

g-Space supports tiff, png, jpg, bmp formats coupled with world files. World files contain information about X and Y for each pixel of the image.
World file has 6 parameters:

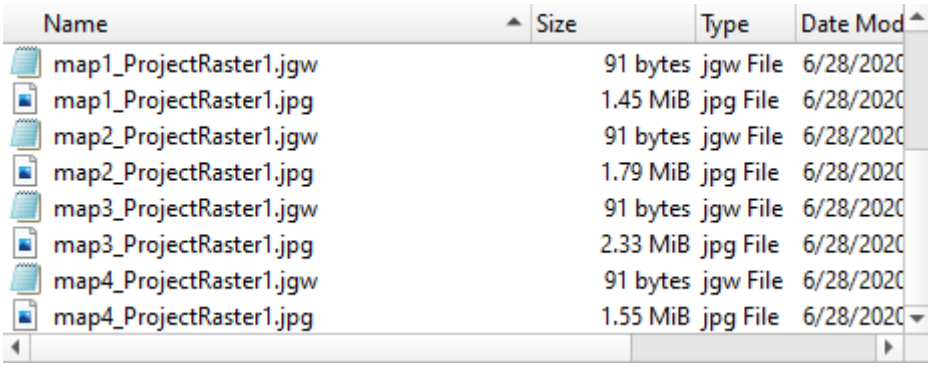
- Line 1: *A*: *x*-component of the pixel width (*x*-scale)
- Line 2: *D*: *y*-component of the pixel width (*y*-skew)
- Line 3: *B*: *x*-component of the pixel height (*x*-skew)
- Line 4: *E*: *y*-component of the pixel height (*y*-scale), typically negative
- Line 5: *C*: *x*-coordinate of the center of the original image's upper left pixel transformed to the map
- Line 6: *F*: *y*-coordinate of the center of the original image's upper left pixel transformed to the map

File example:

© 2026 Geomage 2003 LTD

```
38.1073406255
0.0000000000
0.0000000000
-38.1073406255
170003.4049205466
804998.5160054950
```

They must have same names as image files but different extensions and located in the same directory. Extensions have firs two characters of image file extension and additional 'w' at the end.



For more information can be found here World file - Wikipedia.

Raster basemap and satellite imagery (import)

Georeferenced basemap images (GeoTIFF / TIFF / PNG / JPG / BMP) can be loaded as a cultural background. Satellite imagery can also be downloaded directly from web XYZ map tiles and stored as a georeferenced GeoTIFF.

See Import images 446 and Import satellite imagery 449.

Image export

g-Space can write images out as well:

- **GeoTIFF** - 3D post-stack seismic inline/crossline sections as georeferenced raster. See Export seismic to GeoTIFF 496.
- **Workspace / view image** (JPEG) - a rendered 2D / 3D / well-log view screenshot. See Export view image 502.

7.7 GIS Data Formats

This chapter summarizes spatial data formats currently supported by g-Space

---

Shapefile format

© 2026 Geomage 2003 LTD

Format description: https://www.esri.com/content/dam/esrisites/sitecore-archive/Files/Pdfs/library/whitepapers/pdfs/shapefile.pdf

For information about importing this format into g-Space refer to Import shapefile 443

Cultural point data format

this format is **tab delimited**, and has in first line headers of attributes
first 2 columns contain XY coordinates the rest - are attributes

X	Y	UWI	Well name	Status code
236526	549741	392	ADMON 01	O-DH
153386	594304	81	AFIQ 01	O-DH
151649	599226	303	AFIQ 02	G-DH
142141	550678	380	AGUR 01	G-DH

information about importing this format into g-Space refer to Import shapefile 443

CPS-3 line format

file format example:

```
FFASCI 0 1 "LINES" 0 1e+10
FFATTR 0 1
->polygon_name
193859.269884 594222.387161 0.000000
189181.677974 599692.282867 0.000000
175939.481160 609182.222405 0.000000
162367.876464 615047.532259 0.000000
156438.534607 615970.165270 0.000000
138057.574849 587170.834864 0.000000
159600.850264 572145.097262 0.000000
160457.310755 567268.322777 0.000000
196692.177660 569047.686440 0.000000
195703.954017 583150.791032 0.000000
193859.269884 594222.387161 0.000000
```

For information about importing this format into g-Space refer to Import polygons 444

Point-set / point data format

Point sets and cultural / custom shape points are loaded from delimited text (CSV / TAB / TXT) through the column-mapping import wizard. See Point Data Formats 523 and Import points 460.

© 2026 Geomage 2003 LTD

7.8 Grid Data Formats

Eclipse Grid Format in g-Space

Eclipse format (.dat) is used to store reservoir simulation grid data. The format consists of:

- **.dat file** → Contains grid dimensions, cell properties (porosity, permeability, etc.).
- **.GRDECL file** → Stores corner-point geometry for grid cell coordinates.

g-Space supports importing 452 and exporting 484 **Eclipse grids**, allowing seamless integration with reservoir modeling and simulation workflows.

Note

On export, a tetrahedral grid is resampled to corner-point geometry. Other 3D model and geobody exchange formats (GOCAD SGrid / TSolid / Voxet, RESQML EPC, VTK, DXF) are described in Model Data Formats 522.

7.9 GOCAD Data Formats

g-Space exchanges triangulated surfaces and polygons with GOCAD and SKUA-GOCAD through ASCII files. This is file-level geometry exchange; g-Space does not open a complete GOCAD or SKUA project.

Supported Formats

- **GOCAD TSurf (.ts)** — import and export of triangulated Multi-Z surfaces; export of triangulated fault surfaces.
- **GOCAD PLine (.pl)** — import and export of open or closed polygons and polylines.

GOCAD TSurf

TSurf files describe surface geometry with vertices and triangular facets. g-Space reads `GOCAD TSurf` objects, `VRTX` or `PVRTX` vertices, and `TRGL` triangles. Multiple TSurf objects in one file are imported as separate surfaces. Multiple `TFACE` sections in one object are combined into one surface.

Import a TSurf surface

1. In Data Manager 124, expand **Multi-Z surfaces**.
2. Right-click the destination **Time** or **Depth** folder and select **Import from GOCAD TSurf**.
3. Select one or more `.ts` files and open them.

© 2026 Geomage 2003 LTD

4. Review the import report. It lists each imported surface, its vertex and triangle counts, renamed duplicates, skipped records, and failed files.

If `AXIS_NAME` or `AXIS_UNIT` identifies a time or depth domain that differs from the selected folder, g-Space asks whether to use the domain declared by the file. When the domain is not declared, the selected folder determines the imported domain.

Export a TSurf surface

Right-click a Multi-Z surface and select **Export to GOCAD TSurf**. If the object contains picked sticks but no facets, g-Space triangulates the sticks before export. At least three suitable stick points are required.

To export a fault surface, expand a fault in Data Manager, right-click its surface child, and select **Export to GOCAD TSurf**. Enter the required datum and choose the output file. This command exports the displayed triangulated fault surface, not the original fault-stick representation.

TSurf coordinate and data handling

- **Z direction:** g-Space stores surface Z positive downwards, so depth and time values increase downwards. A file that declares `ZPOSITIVE Depth`, or that declares no `ZPOSITIVE` record at all, is imported with its Z values unchanged: a vertex stored at 500 m appears at the 500 m depth level, alongside wells and depth surfaces. `ZPOSITIVE Elevation` values are negated on import, so a file that holds elevations but omits the record is imported inverted. Export declares `ZPOSITIVE Depth` and keeps the same downward-positive sign, so an exported surface re-imports at its original position.
- **Coordinate reference system:** TSurf coordinates are imported as stored. g-Space does not transform them from the coordinate-system block. Verify that they match the project CRS before import.
- **Units:** axis units declared in the file are converted to the project units on import; missing axis units produce a warning and are assumed to be metres. Export declares the units of the current project measurement system: X and Y in the project distance unit, and Z in the project depth unit for a depth surface or in milliseconds for a time surface.
- **Properties:** trailing values on `PVRTX` records and other surface properties are not retained. Only XYZ geometry and triangle connectivity are exchanged.
- **Names:** the TSurf header name becomes the surface name. If that name already exists, g-Space adds a numeric suffix.

GOCAD PLine

PLine files describe connected line geometry. g-Space reads `GOCAD PLine` objects, `ILINE` sections, `VRTX` or `PVRTX` vertices, and `SEG` connections. Connected loops become closed

© 2026 Geomage 2003 LTD

polygons; open chains remain open polylines. Disconnected segment chains are imported as separate polygon objects.

Import polygons from PLine

1. In Data Manager 124, right-click the **Polygons** root.
2. Point to **Import** and select **Import polygons from GOCAD PLine file**.
3. Select one or more `.pl` files, then specify the source coordinate system and measurement system in the import dialog.
4. Review any import report warnings. Invalid records are skipped when valid geometry can still be imported.

Export polygons to PLine

Right-click an individual polygon and select **Export to GOCAD PLine file**, or right-click a polygon folder and select **Export all polygons to GOCAD PLine file**. g-Space writes each polygon as a GOCAD PLine object with one `ILINE`. Closed polygons include a final segment connecting the last vertex to the first.

PLine limitations

- Only XYZ coordinates, names, segment connectivity, and open or closed state are retained.
- Per-vertex `PVRTX` properties and property metadata are discarded during import.
- A PLine with vertices but no `SEG` records is imported as one open polyline in vertex order.
- Malformed vertices or segments generate warnings and are skipped. A file with no valid line geometry is rejected.

See Also

Data Manager 124 · Fault Data Formats 515 · GIS Data Formats 517 · Model Data Formats 522

7.10 Trace headers math

Here is located information about available functions in the trace header expressions

In this module the user can able to perform many mathematical operations to modify the trace headers reading.
User can use any of the following mathematical operations to create their own equation/expression.

To reference to the trace field use "data type (byte)". For example, the expression "int4(185)" returns the value read as a 4-byte integer from byte number 185.

© 2026 Geomage 2003 LTD

g-Space uses following mathematical expressions in designing your equation:
Mathematical operators (+, -, *, /, %, ^);
Functions (min, max, avg, sum, abs, ceil, floor, round, roundn, exp, log, log10, logn, root, sqrt, clamp, inrange);
Trigonometry (sin, cos, tan, acos, asin, atan, atan2, cosh, cot, csc, sec, sinh, tanh, d2r, r2d, d2g, g2d, hyp);
Equalities, Inequalities (=, ==, <>, !=, <, <=, >, >=);
Assignment (:=, +=, -=, *=, /=);
Boolean logic (and, nand, nor, not, or, xor, xnor, mand, mor);
Control Structures (if-then-else, ternary conditional, switch case);
Loop Structures (while loop, for loop, repeat until loop, break, continue)

7.11 Model Data Formats

Model Data Formats are the exchange formats g-Space uses for 3D models, geobodies and volumes - structured and tetrahedral grids, voxel volumes, and geobody geometry. Triangulated surfaces and polylines are described separately in GOCAD Data Formats 519; corner-point reservoir grids in Grid Data Formats 519.

Supported formats

- **GOCAD SGrid** (ASCII .sg + companion data) - 3D structured grid with grid properties. Import and export.
- **GOCAD TSolid** - tetrahedral grid. Import and export.
- **GOCAD Voxet** - 3D voxel grid. Import only.
- **RESQML EPC** (.epc) - polygons, tetrahedral / 3D grids, multi-Z surfaces and point sets. Import and export.
- **VTK** (.vtk) - tetrahedral grid as an unstructured mesh with properties. Export only.
- **DXF** (AutoCAD, .dxf) - polygons and geobodies. Import and export.

GOCAD SGrid, TSolid and Voxet

GOCAD SGrid stores a 3D structured (stratigraphic) grid together with its grid properties; it is written as an ASCII header (.sg) plus companion binary/ASCII data files. **GOCAD TSolid** stores a tetrahedral grid, and **GOCAD Voxet** stores a regular 3D voxel volume. SGrid and TSolid support both import and export; Voxet is import only.
For the workflow see Import 3D models 453 and Export 3D models 485.

© 2026 Geomage 2003 LTD

RESQML EPC

RESQML is the Energistics reservoir data-exchange standard. A single `.epc` package can carry polygons, tetrahedral and 3D grids, multi-Z surfaces and point sets. Both import and export are supported.
See Import RESQML 456 and Export RESQML 486.

VTK

A tetrahedral grid can be exported to **VTK** (.vtk) as an unstructured mesh with its properties, for use in external visualization tools. VTK is export only. See Export to VTK 488.

DXF (AutoCAD)

DXF files carry polygons and geobody geometry. g-Space reads DXF through the bundled libdxfrw library and offers an option to invert the Z axis on import. Both import and export are supported.
See Import DXF 458 and Export DXF 489.

See Also

Grid Data Formats 519 | GOCAD Data Formats 519 | Import grids 452 | Export grids 484

7.12 Point Data Formats

Point Data Formats cover point sets and cultural / custom shape points. These are loaded from plain delimited text through the shared column-mapping import wizard, where you assign each text column to a field and choose the target object type. Polygons and line geometry are described in GIS Data Formats 517 and GOCAD Data Formats 519.

Supported formats

- **Point-set / point-data ASCII** (CSV / TAB / .txt) - point sets and cultural / custom points. Import and export.
- **Cultural / custom shape points text** (CSV / TAB / .txt) - cultural points and custom shape points. Import only.
- **GOCAD PLine** (.pl) - polygons / polylines (see GOCAD Data Formats).

© 2026 Geomage 2003 LTD

File layout

The file is plain delimited text. The first two columns hold the X and Y coordinates; the remaining columns are attributes. When a header line is present, its names are offered as field labels in the import wizard.
The same *column-mapping import wizard* underlies most ASCII imports in g-Space (point sets, well headers, markers / tops, inclinometry, horizons, fault sticks). The format is always plain delimited text; the object type is chosen in the wizard.

See Also

Import points 460 | Export points 499 | GIS Data Formats 517 | GOCAD Data Formats 519

© 2026 Geomage 2003 LTD

Service Tasks

8 Service Tasks

Service tasks are the long-running background operations that g-Space schedules on behalf of the user — for example recalculating project parameters after new data is loaded, synchronising views, or executing a multi-step procedure. They are controlled from the Main bar[35] of the ribbon and their progress is reported in the Output View[219].

Execute process

Some data-loading operations only register objects in the project; a follow-up recalculation is required to integrate the new objects with existing horizons, wells, or maps. Trigger this recalculation by opening the Ribbon[33] → Main bar[35] and clicking **Execute**. g-Space runs every pending procedure in the queue and writes the results back to the project.

Stop process

If a running task takes longer than expected or was launched by mistake, use Ribbon[33] → Main bar[35] → **Stop**. The current task is aborted cleanly — partial results are discarded and the project returns to the state it was in before the task started.

Monitoring progress

Task status, elapsed time and log messages appear in the Output View[219] dock panel while a service task runs. Warnings and errors raised by the underlying procedure are recorded there too, so the panel is the primary place to check when a task ends unexpectedly.

See Also

- Main bar[35]
- Output View[219]
- Ribbon bar[33]

© 2026 Geomage 2003 LTD

Best Practice Flows

9 Best Practice Flows

g-Space software offers a comprehensive suite of tools for seismic interpretation and geological modeling, enabling users to efficiently analyze and model complex subsurface structures. To help users maximize the potential of g-Space, this section of the manual outlines best practice workflows, illustrated through a demo project based on the Teapot Dome oil field.

Teapot Dome, located in the Powder River Basin, Wyoming, is a geologically significant field known for its complex structural features and productive hydrocarbon reservoirs. The field primarily produces from the Cretaceous and Pennsylvanian formations, with key reservoirs including the Tensleep Sandstone and the Frontier Formation. The anticline structure of Teapot Dome, formed during the Laramide orogeny, plays a crucial role in trapping hydrocarbons, while regional seals such as the Mowry Shale ensure the accumulation and preservation of these resources.

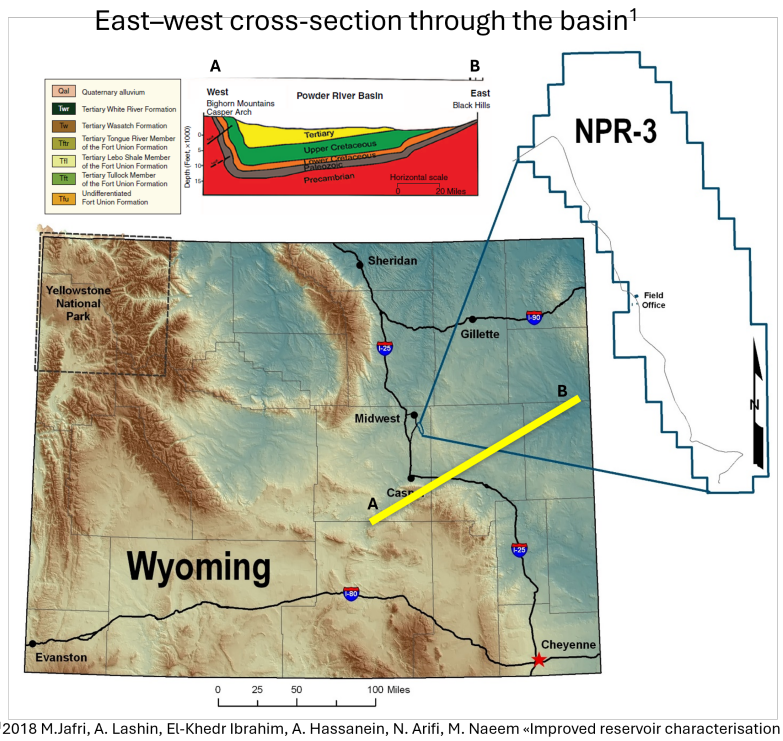
All data available to the public and can be downloaded here https://wiki.seg.org/wiki/Teapot_dome_3D_survey

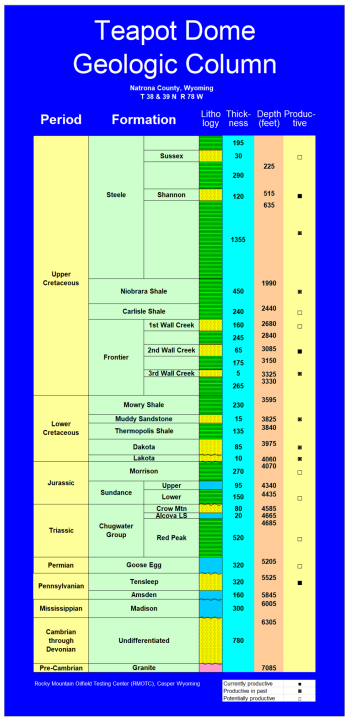
This demo project will guide you through the essential steps of seismic interpretation and geological modeling within g-Space, using real data from the Teapot Dome field. By following these best practices, users can effectively build accurate geological models, optimize their workflows, and make informed decisions for exploration and production activities.

When using this manual, remember that the methods in the Help section are tailored to a specific example with a particular dataset. Each geological model is different, with unique data and targets. So, the steps and methods you use may vary. By completing the correlation in the Demo project, you'll get to know the g-Space software interface and learn modeling techniques suited to the training data.

© 2026 Geomage 2003 LTD

East–west cross-section through the basin[1]





[1]2018 M.Jafri, A. Lashin, El-Khedr Ibrahim, A. Hassanein, N. Arifi, M. Naeem «Improved reservoir characterisation using fuzzy logic platform: an integrated petrophysical, seismic structural and poststack inversion study»

Input data

The following data will be used as initial inputs for seismic interpretation and geomodeling:

- 2D and 3D seismic data (all data in feet, SRD=5500 ft above sea level, Vrep=9000 ft/s)
- Results of seismic data interpretation (horizons, faults)
- Well data (wellheads, inclinometry, log data, checkshots, markers)
- Topography
- Boundaries

Please note that all data in the Demo Project is presented in the imperial system of units, with feet as the base unit of measurement. Project coordinate system is 32056 NAD27 / Wyoming East Central.

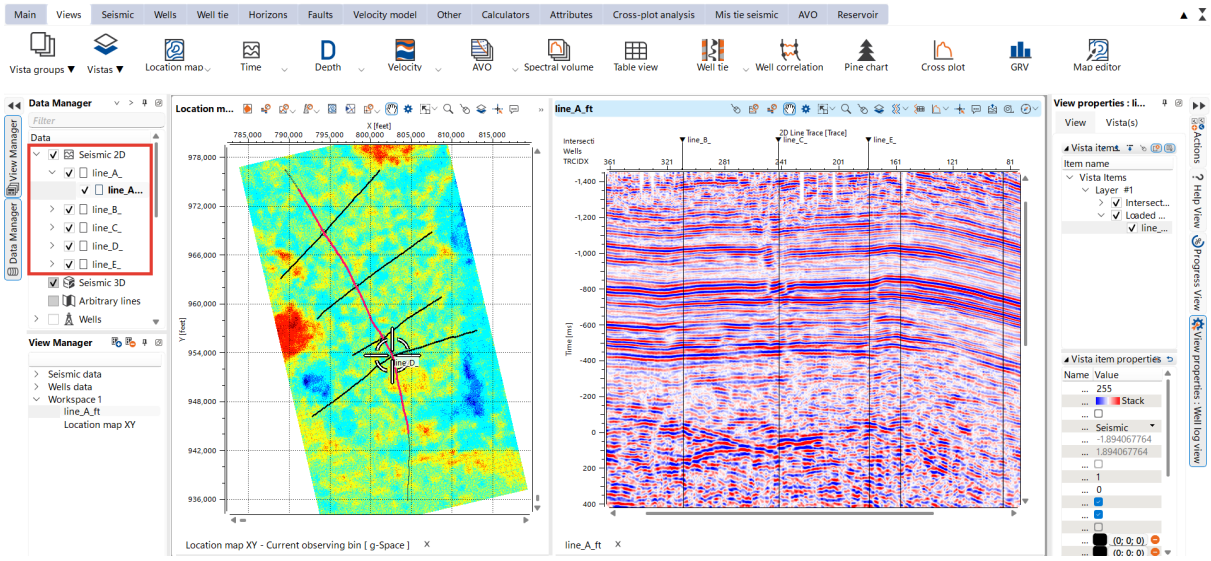
The following seismic information will be used in the project

2D seismic profiles in SGY-format:

- line_A_ft
- line_B_ft
- line_C_ft

© 2026 Geomage 2003 LTD

- line_D_ft
- line_E_ft

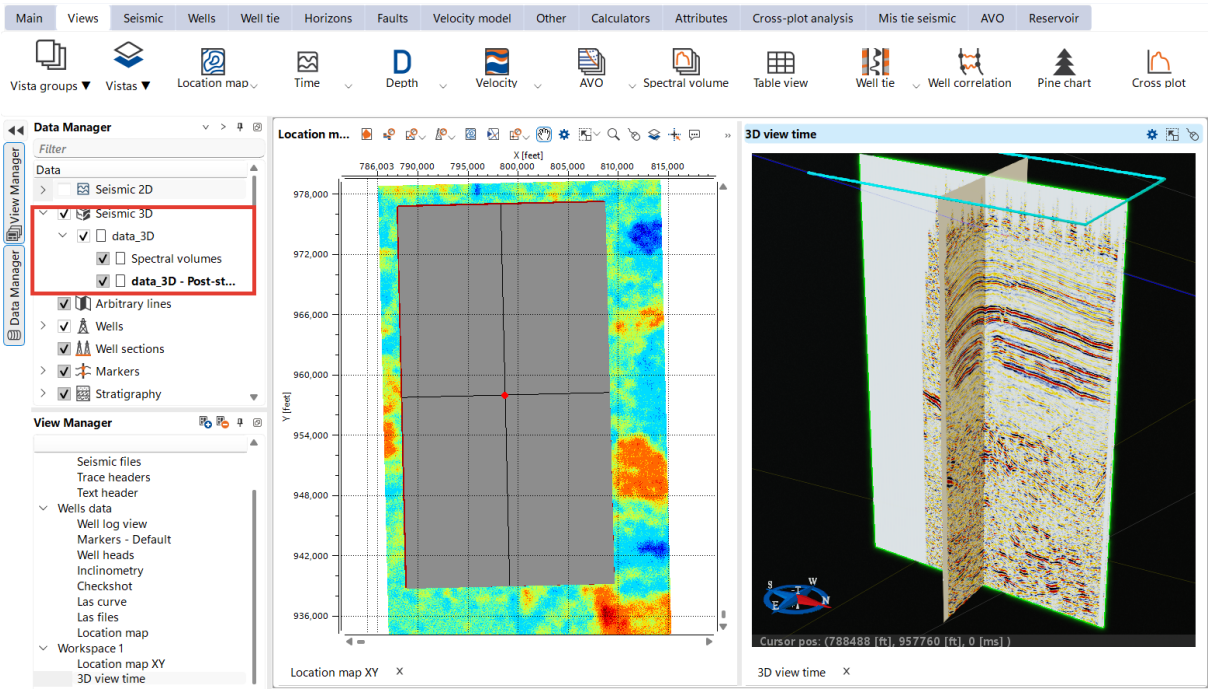


3D seismic volume in SGY-format:

- data_3D_teapot_ft

Inline located in byte 189
Crossline located in byte 193
X coordinate located in byte 181
Y coordinate located in byte 185

© 2026 Geomage 2003 LTD



There are existed horizons and faults interpretation in the project.

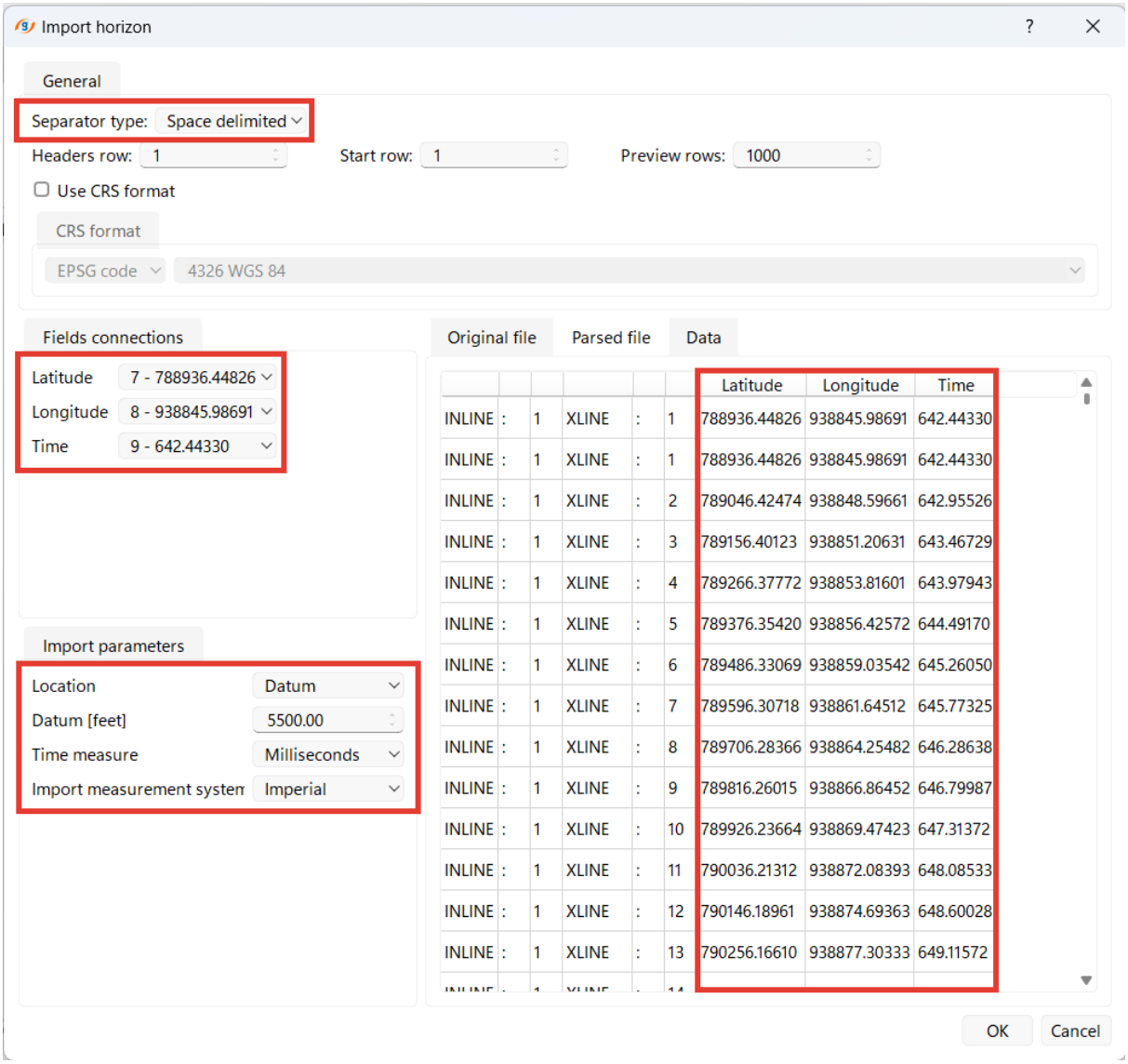
Horizons are presented as text files with inline and crossline numbers and XYZ coordinates.

- Carlile_Shale
- Frontier_2_WC
- Lakota_Sand
- RedPeak_Shale
- SRD_5500_TimeZero
- Tensleep_Sand
- Topo_Time

An example of the file is provided below.

Please be mindful of the system of units and measurement units as well as Datum level

© 2026 Geomage 2003 LTD

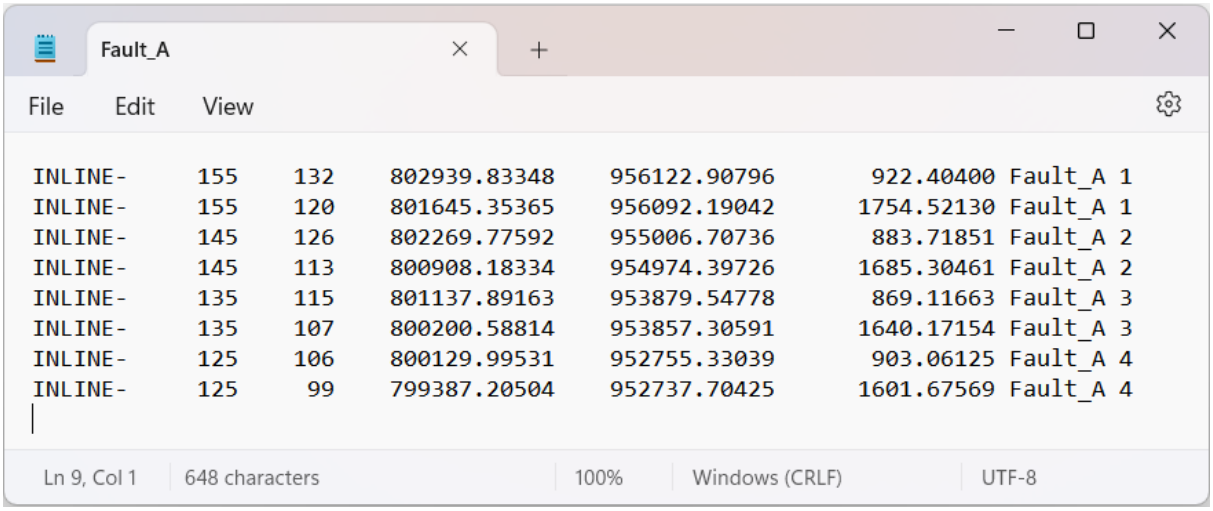


Faults are presented as text files with inline and crossline numbers, XYZ coordinates, Name and Sticks number (Charisma Stick Format):

- Fault_A
- Fault_B
- Fault_C
- Fault_B

An example of the file is provided below.

© 2026 Geomage 2003 LTD

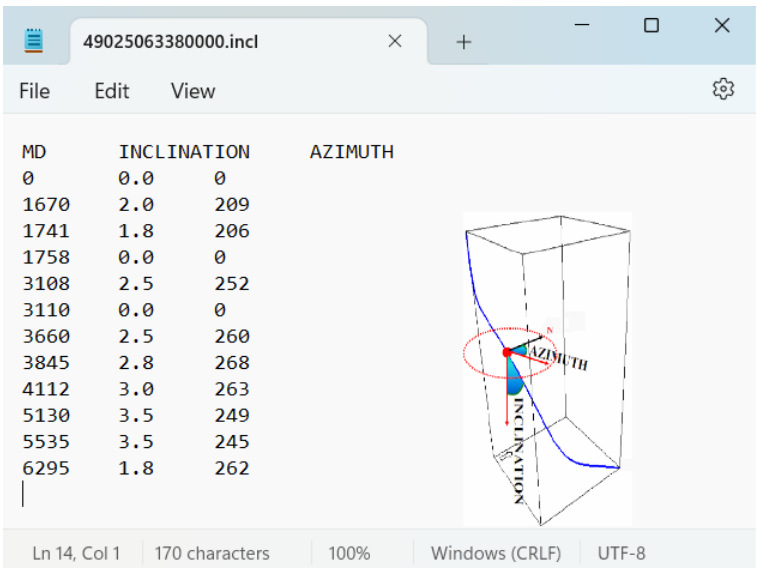


The Demo Project includes 36 wells, each accompanied by wellhead data, inclinometry, log data, and markers. For your convenience, the wells are categorized into those where acoustic and density logging has been performed and those where it has not.

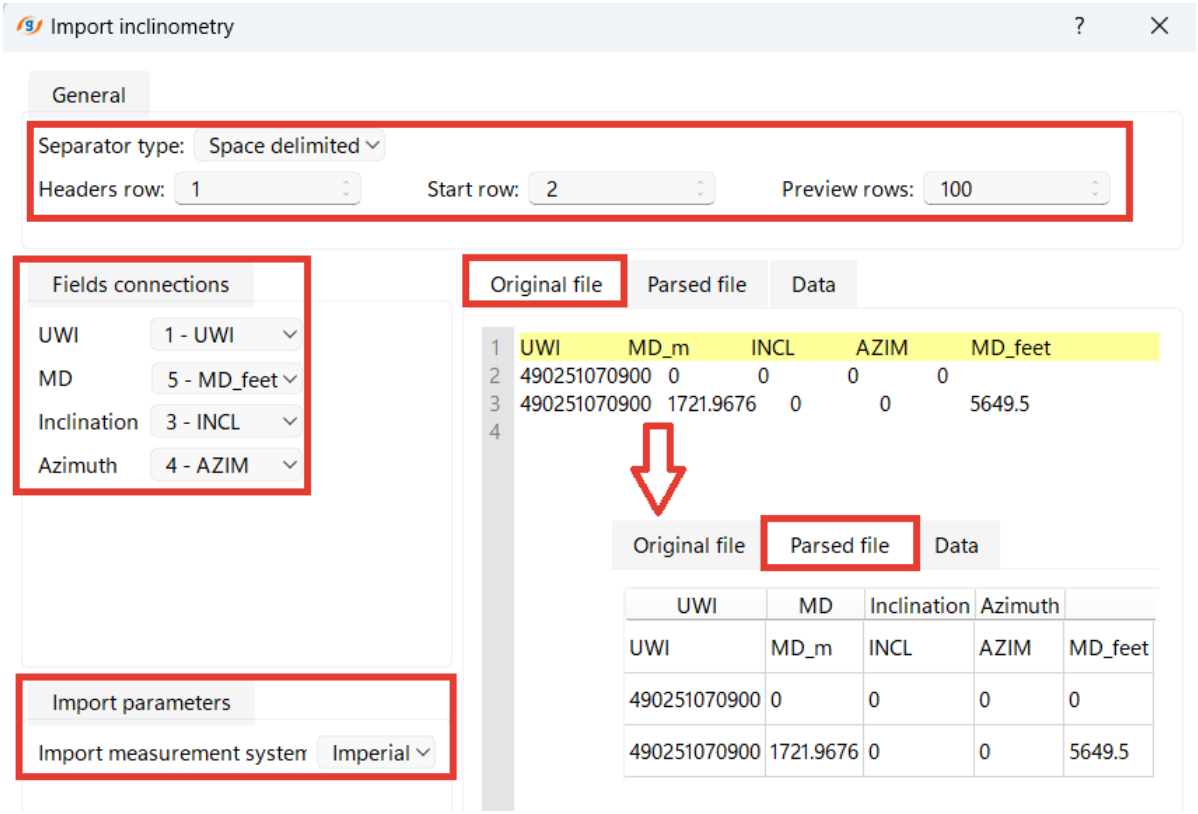
The well head data is in Well_heads.prn. Use Import well headers 393 to set the reading parameters and map the separate **Type** and **Status** fields.

All Coordinates in Wyoming East Central State Plane, NAD 1927

Well inclinometry is presented in the demo project as a set of files in the standard format *incl and includes 2 folders: Wells with long P-wave & Density and Others



© 2026 Geomage 2003 LTD



Easily check inclinometry data in the pre-installed Wells Data Workspace 232, where you can visualize all well data.

Log data for wells is presented in the form of standard LAS files and contains logging curves including density and acoustic.

Well markers are provided in text format and contain well names, marker names, X, Y, Z and MD.
Below is a list of marker names:

- Shannon_Up
- Steele_B
- Steele_F
- SteeleGD
- Niobrara_WS
- Carlile_Shale
- Frontier_2_WC
- Mowry_Shale
- Dakota_Sand
- Lakota_Sand

© 2026 Geomage 2003 LTD

- Crow_M_Sand
- RedPeak_Shale
- Tensleep_Dolo
- Tensleep_Sand

Time-depth curves are available for 10 wells in the Demo Project.

Use the installed Wells data workspace 232 to review the imported well data.

The database also includes a polygon with the project boundaries and topography data.

For data uploading, please use Import data 373 instructions.

9.1 Workflows

A **workflow** in g-Space is a named sequence of jobs — procedures and database operations — that run together as a single chain. Workflows are stored inside the project so they are available to every user with access to the project and can be re-executed without re-entering parameters.

Creating and managing workflows

- **Create** — add a new workflow with a user-supplied name.
- **Rename** — change the name of an existing workflow.
- **Remove** — delete a workflow from the project.
- **Set current** — choose which workflow is edited and executed next.

Building a workflow

Each workflow is composed of **jobs** and, optionally, **notes**. A job wraps one g-Space procedure plus its saved parameters, or a database operation; a note is a comment row that documents the sequence but never runs. Jobs can be added, renamed, reopened for editing, reordered and removed inside the active workflow. Execution runs the jobs sequentially, feeding the output of one into the next when their data types line up.
Progress is reported in the Output View 219 while the workflow runs. If any job fails, execution stops and the workflow can be resumed after the error is corrected.

Adding jobs from a wizard

© 2026 Geomage 2003 LTD

Most g-Space procedure wizards can store their current parameters as a workflow job instead of (or as well as) running immediately. When a wizard supports this it shows a **Save to workflow** button and its standard buttons are relabelled:

- **Run** — execute the procedure now with the current parameters. The wizard stays open, so parameters can be adjusted and the procedure re-run.
- **Save to workflow** — add the current parameters as a new job in the active workflow. The wizard stays open and the entered parameters are kept, so a job can be saved and then run, or several jobs saved in turn.
- **Close** — close the wizard. Closing does not apply or save the parameters.

If the project does not contain any workflow yet, choosing **Save to workflow** offers to create one on the spot: enter a name for the new workflow and the job is added to it, without a separate trip to the Workflows wizard.

These labels apply to a wizard opened to configure a new job or to run the procedure. A wizard opened from a job that is already stored in a workflow shows a different set of buttons — see Editing a saved job below.

Adding notes to a workflow

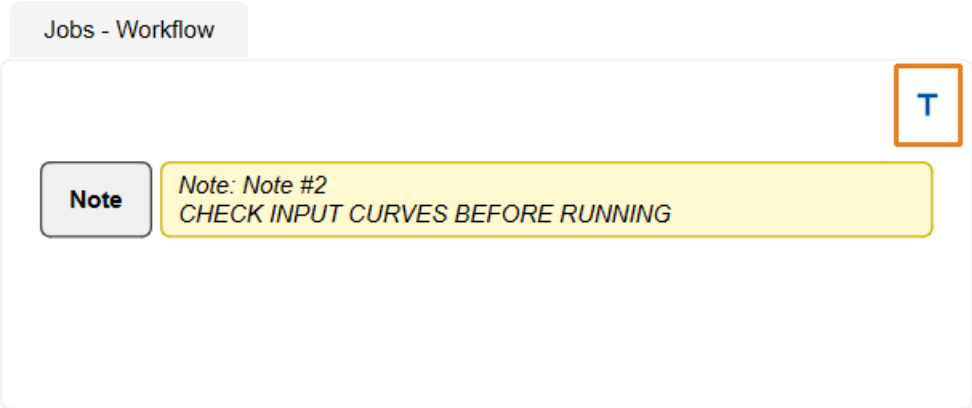
A workflow can also carry **notes** — short comments that explain the sequence to whoever runs it next. In the **Jobs** panel of the Workflows wizard, click the note button above the job list (its tooltip reads **Add note**), type the text in the **Add note** dialog and confirm. The note is appended to the current workflow as its own row. If the project does not contain any workflow yet, the same **Create workflow** prompt described above appears first.

A note row is marked with a **Note** badge in place of the **Start** or **End** badge, and is labelled **Note:** followed by the note's name, with a one-line preview of the text below it. A long text is shortened in the preview and a note left empty is shown as **(empty note)**; point at the row to see the full text.

To change the text, double-click the note row, or right-click it and choose **Edit**, and edit the text in the **Edit note** dialog. Notes are renamed and removed with the same **Rename** and **Remove** commands as jobs.

Notes never execute. A note placed between two jobs passes the data chain through untouched, so it does not change what the workflow produces. Notes are stored with the workflow and are restored when the project is opened again.

© 2026 Geomage 2003 LTD



Editing a saved job

A job that is already stored in a workflow can be reopened and edited. In the **Jobs** panel of the Workflows wizard, right-click the job and choose **Open**, or double-click the job. A single left-click does not open the job: it drags the job to a new position in the sequence. The same right-click menu also offers **Move up** and **Move down** as an alternative to dragging; **Move up** is unavailable on the first row and **Move down** on the last. On a note row **Open** is replaced by **Edit**, and double-clicking opens the note text instead of a wizard.

Open reopens the procedure's own wizard with the parameters stored in the job already filled in. Most of these wizards open as a single dialog; while a saved job is being edited there, **Run** is not offered and the wizard has two buttons:

- **Save** — write the edited parameters back into the same job and close the wizard. The job keeps its name, its position in the sequence and any **Start** or **End** badge.
- **Cancel** — close the wizard and leave the job unchanged.

To run the procedure with the edited parameters, save the job and then execute the workflow.

A wizard that is stepped through page by page with **Back** and **Next** — for example **Create velocity model** — keeps its usual **Save to workflow**, **Run** and **Close** buttons when a saved job is reopened. **Save to workflow** writes the edited parameters back into the same job; **Run** carries out the procedure immediately with the current parameters and leaves the stored job unchanged.

Setting Start and End jobs

A workflow can be executed in full or restricted to a contiguous range of its jobs. In the **Jobs** panel of the Workflows wizard, right-click a job and choose:

© 2026 Geomage 2003 LTD

- **Set as Start** — mark the job as the first job to run. A **Start** badge appears beside it.
- **Set as End** — mark the job as the last job to run. An **End** badge appears beside it.
- **Clear Start** / **Clear End** — remove the marker so execution reverts to the first or last job.

When the workflow runs, execution begins at the Start job (or the first job when no Start is set) and stops after the End job (or the last job when no End is set). The Start job cannot come after the End job: setting a Start that is later than the current End clears the End marker, and setting an End that is earlier than the current Start clears the Start marker. Start and End can be set on any existing workflow and are stored with it. These commands are offered on job rows only: a note never executes, so it cannot be a run boundary.

Executing a workflow

1. Select the workflow to execute in the workflows wizard.
2. Confirm the parameters of every job — double-click a job, or right-click it and choose **Open**, to inspect or change its saved parameters. Doing the same on a note row opens the note text for editing and does not affect the run.
3. Press **Execute**. The full chain runs as a single service task, with progress reported in the Output View.

When to use

- Repeatable workflows used across many wells or survey lines (for example standard petrophysical stacks).
- Team-wide protocols where the same sequence of procedures must be applied to every project.
- Pipelines chaining import, processing and export operations without intermediate user input.

See Also

- Python scripting 811
- Output View 219
- Service Tasks 526

9.2 Well log correlation

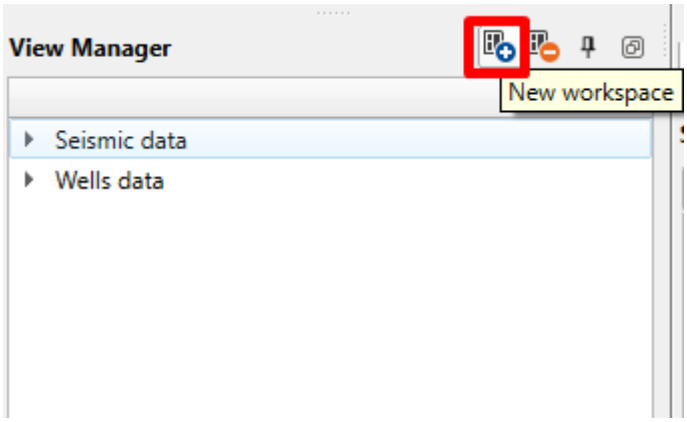
This chapter will help you get started with well correlation.

© 2026 Geomage 2003 LTD

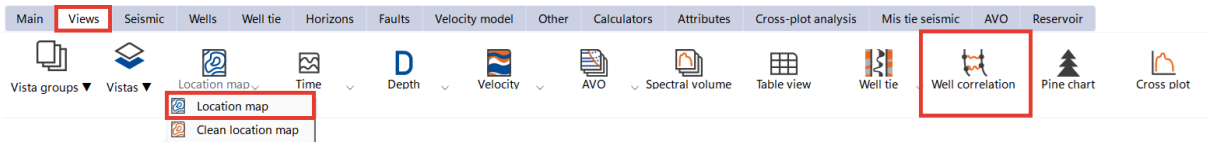
Subscribe to our channel and watch the video on how to correlate wells in g-Space on YouTube

Information about wells data loading you can find here 391.

Once the data has been successfully loaded, proceed to the View manager 189 panel. Create a new workspace and assign it a suitable name:

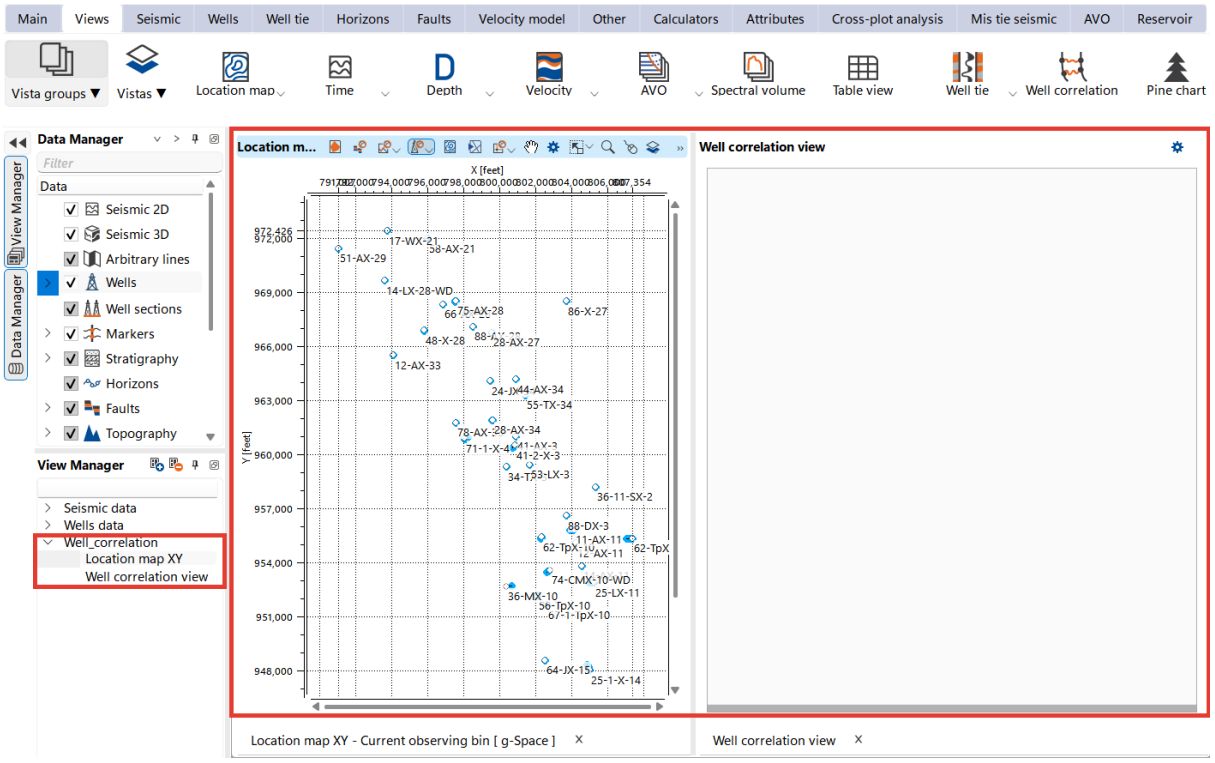


Create two views: Location map and Well correlation

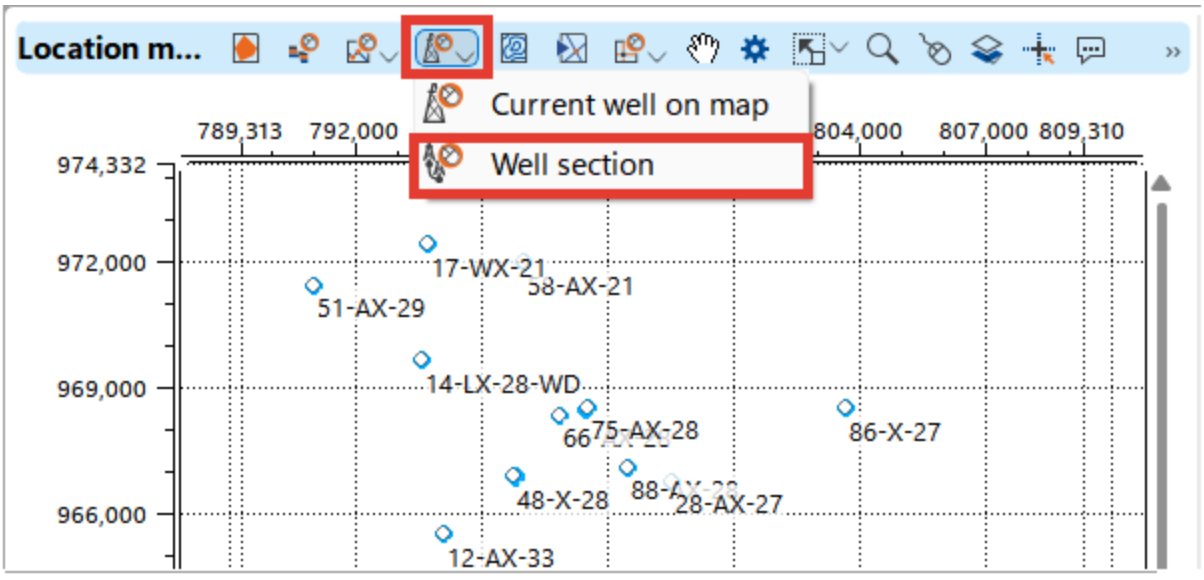


Now there are two views in your workspace

© 2026 Geomage 2003 LTD

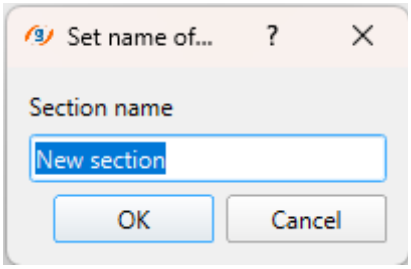


To create a Well correlation section, click the Well button on the tool panel in the Location map view and select Well section.



On the first well, g-Space creates a well section and prompts you to name it

© 2026 Geomage 2003 LTD



Choose the wells you want to add to your section. The selected wells will appear in the **Well correlation view**. This workspace is where the core of your correlation work takes place, allowing you to visually compare and align well logs, stratigraphic markers, and other key data across multiple wells.

The simplest way to build a section is directly from the **Data Manager**: check the box next to a well. If no well section exists yet, g-Space prompts you to create one and name it. Each additional well you check is then added to the active well section.

Effective well correlation often requires tailoring the display settings to highlight the most relevant data for your analysis. Here's how you can customize your Well Correlation view:

Accessing Visualization Settings:
To adjust the appearance and properties of any object in the correlation window, simply right-click on the object. This will bring up a menu of visualization options where you can modify aspects such as color, scale, and labeling.

Settings are organized in three levels, each reached by right-clicking a different area:

- **Global** — right-click the well name at the top of the well section.
- **Track** — right-click the top of a track.
- **Curve** — right-click the curve header.

Let's examine the workspace of the Well correlation view.

1. **Managing Well Sections in Data Manager:**

As you create and modify well sections, they will automatically appear in the Data Manager. This allows you to keep track of your sections and manage them alongside other project data.

2. **Adjusting Axis Settings:**

The axis settings provide control over the visualization of depth, time, and other relevant axes. You can customize parameters such as axis names, scales, colors, and label sizes to ensure the display is clear and informative.

3. **Configuring Track Settings:**

Each track in the correlation view — whether it's a log curve, stratigraphic marker, or another data type — can be customized to suit your analysis. Track settings include options for visualization, adding markers, and adjusting display properties.

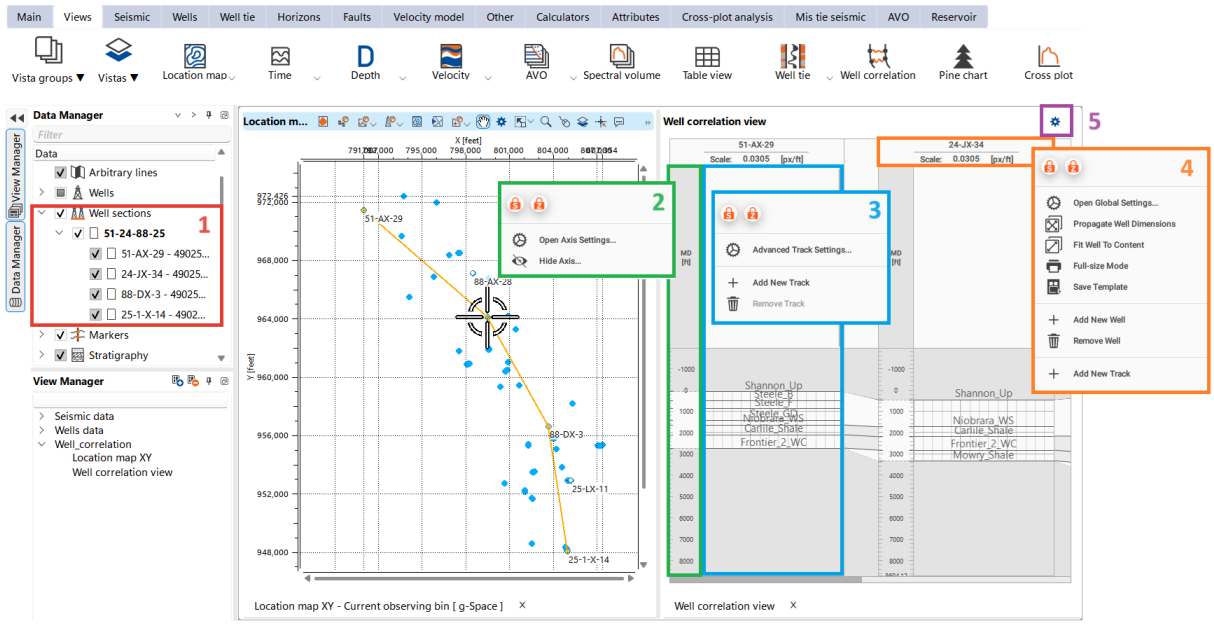
© 2026 Geomage 2003 LTD

4. Saving Well Templates for Future Use:

Once you've configured a well template that fits your needs, save it within the Well settings. This template can be easily recalled in future projects, streamlining your workflow and ensuring consistency across your work.

5. Exploring Global Settings:

The global settings in the Well Correlation view offer a comprehensive suite of tools for refining your correlation. These tools are described in detail within the Well correlation View 296 section of the Help documentation, providing guidance on advanced customization and usage.



Notes

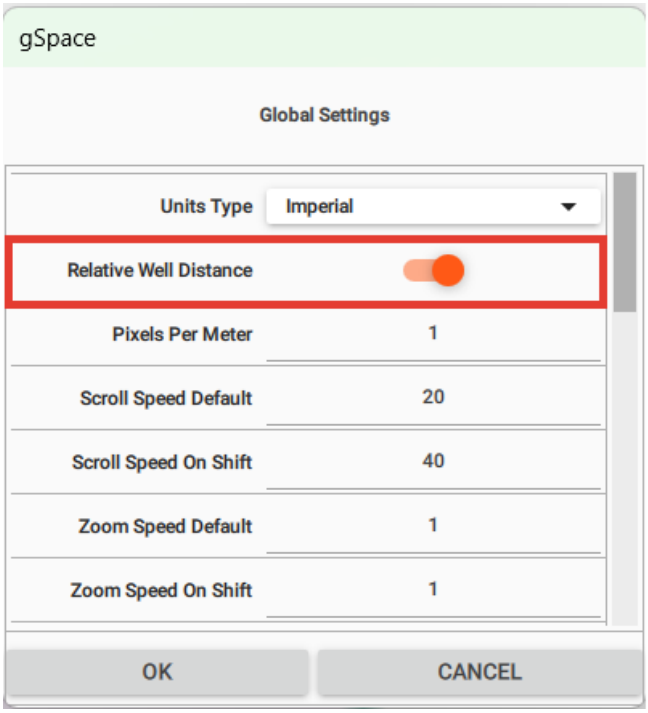
1. *Consider the structural framework of the area, such as faults, folds, or unconformities. These features can disrupt or offset geological layers, making correlation more complex. Recognizing these structures early on can guide your correlation and prevent errors.*
2. *Start your correlation with key wells that have the most reliable and detailed data, such as wells with core samples, high-quality logs, or known stratigraphic markers. These key wells serve as anchors, providing a solid foundation for correlating other wells with less detailed information.*
3. *Use multiple wells to cross-check correlations, especially in complex geological settings. This approach helps to validate the continuity of horizons and reduces the risk of misinterpretation.*
4. *Be aware that well logs from different wells may not always correlate perfectly due to differences in data quality, measurement methods, or geological variations. In such cases, use your geological knowledge to interpret these discrepancies rather than forcing a correlation.*
5. *If available, use seismic data to tie well correlations to broader regional interpretations. Seismic horizons can provide additional context and help validate well correlations,*

© 2026 Geomage 2003 LTD

especially in areas with complex geology. Generating synthetic seismograms from well logs can help bridge the gap between well data and seismic interpretation.

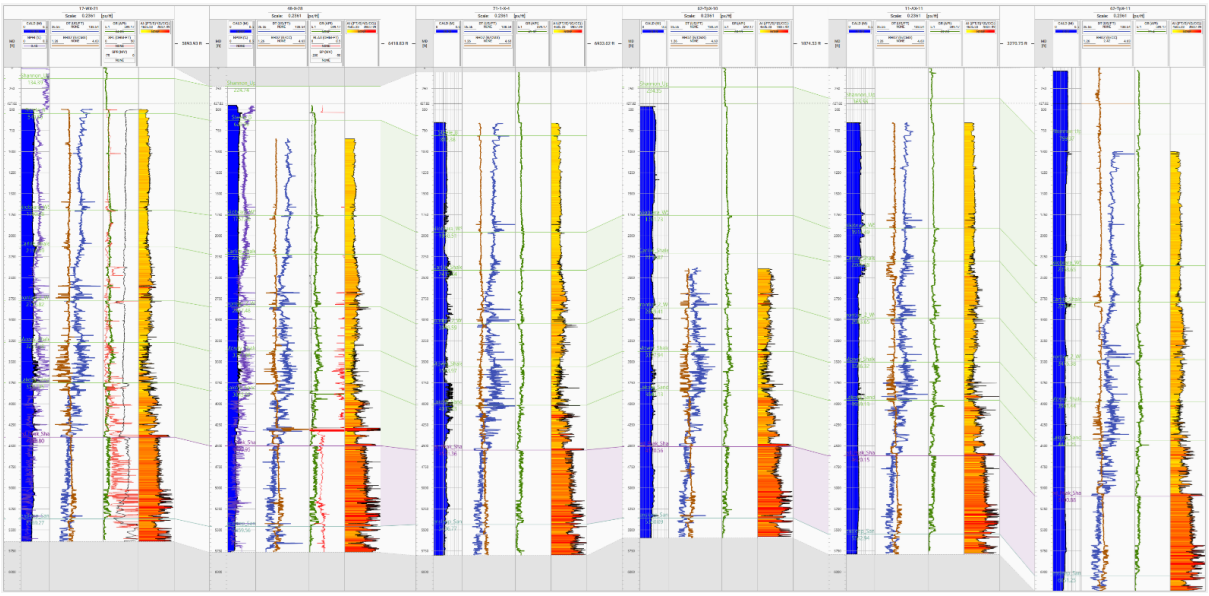
6. *Keep detailed records of your correlation process, including decisions made, marker positions, and any adjustments. This documentation is crucial for future reference and for communicating your findings with others.*

To accurately represent the proportional distance between wells on the correlation diagram, utilize the Relative Well Distance function found in the Global Settings.



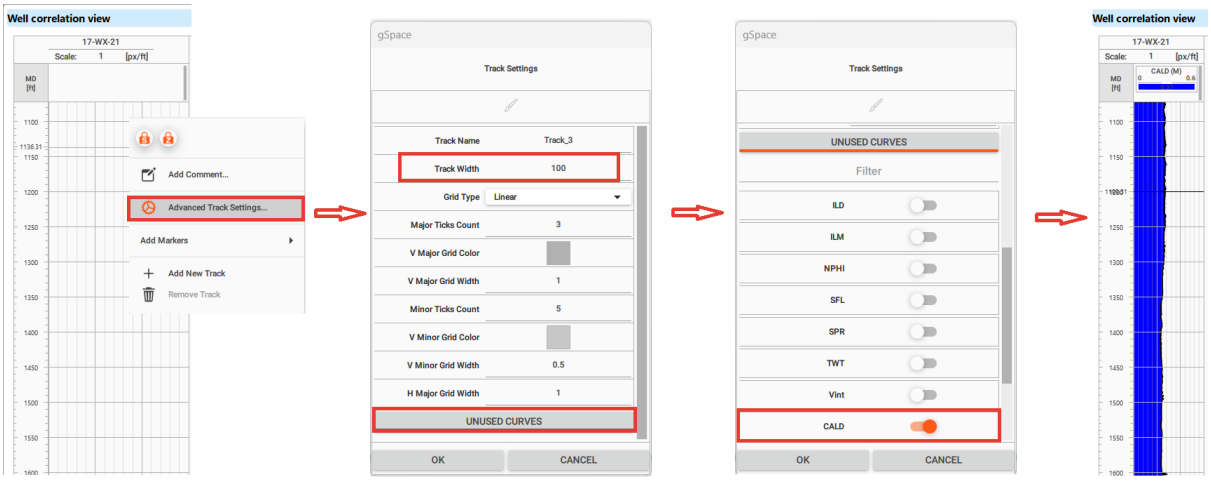
After customizing 544 your Well Correlation view and adding necessary markers 551, review your correlation to ensure accuracy and consistency across all wells.

© 2026 Geomage 2003 LTD



9.2.1 Setting Up a Well Template

To visualize a curve in an empty track, right-click on the track display and select **Advanced Track Settings**. This will open a settings window. For this example, set the **Track Width** to 100. Scroll to the bottom of the window and click on **Unused Curves** to display a list of curves that haven't been visualized yet. Select the **caliper** curve from the list. The curve will be visualized.



To set up curve visualization, right-click on the curve name field at the top of the panel (highlighted in red). This will open the curve visualization settings window.

© 2026 Geomage 2003 LTD

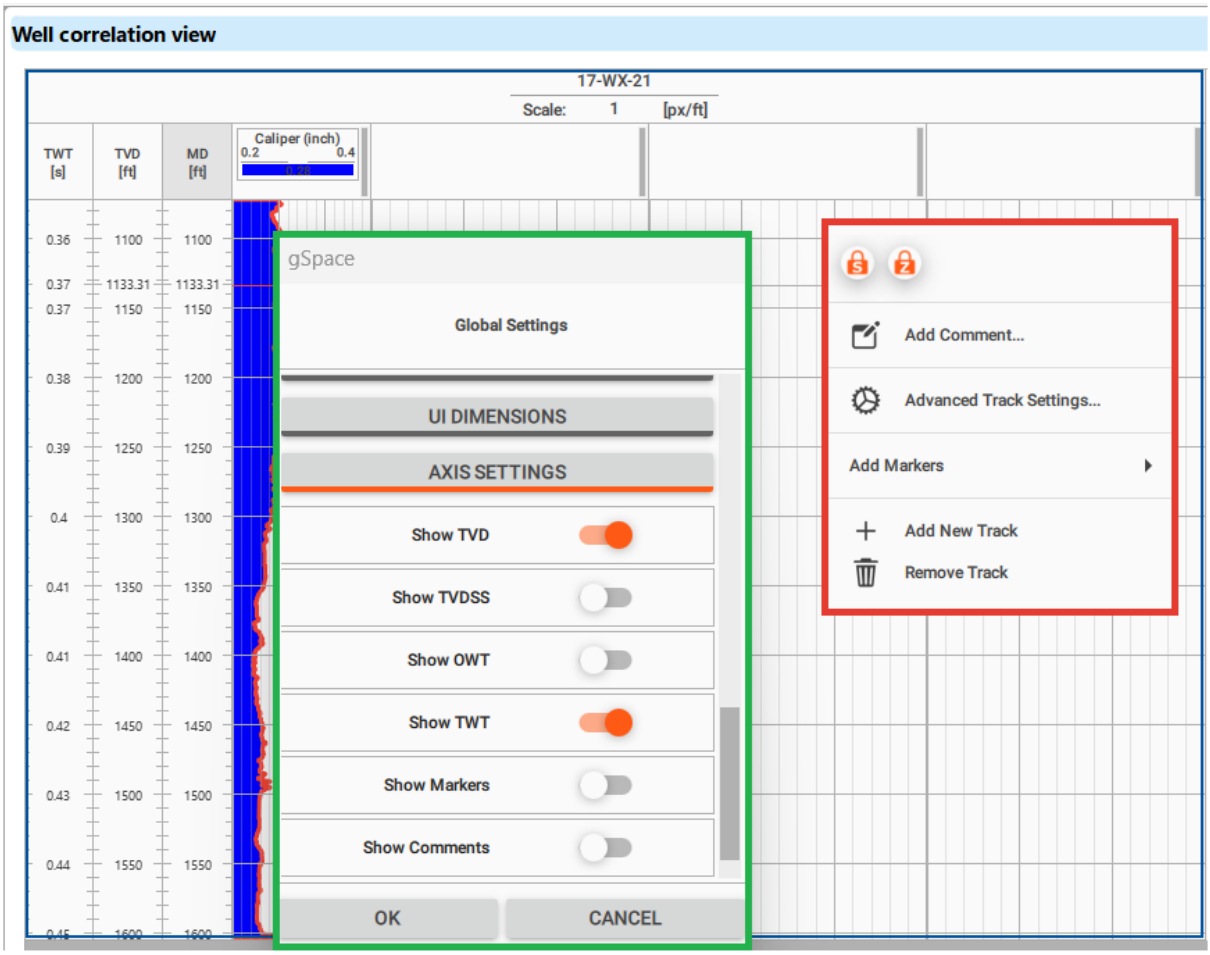
In this example, the following adjustments were made:

- Renamed the **Curve Nickname**
- Updated the **Units**
- Selected a filled **Curve Type** (highlighted in green)
- Selected a **Step Style** (highlighted in blue)
- For a Lines or Filled Lines curve, selected a **Line Style** (Solid, Dot, Dash, Dash dot, or Dash dot dot)
- Set the **Min Value** and **Max Value**
- Chose curve **Color** and **Width**
- Chose **Fill color**

For a current view of the **Curve Settings** panel, including **Line Style** and palette controls, see Working with Palettes 323.

To add three additional tracks for the curves, right-click on the visualized track field and select **Add Track** (highlighted in red). Next, access the global settings (highlighted in green) by right-clicking on the well name at the top.
Scroll down to the **Axis Settings** and enable the visualization of **TVD** (True Vertical Depth) and **TWT** (Two-Way Time).

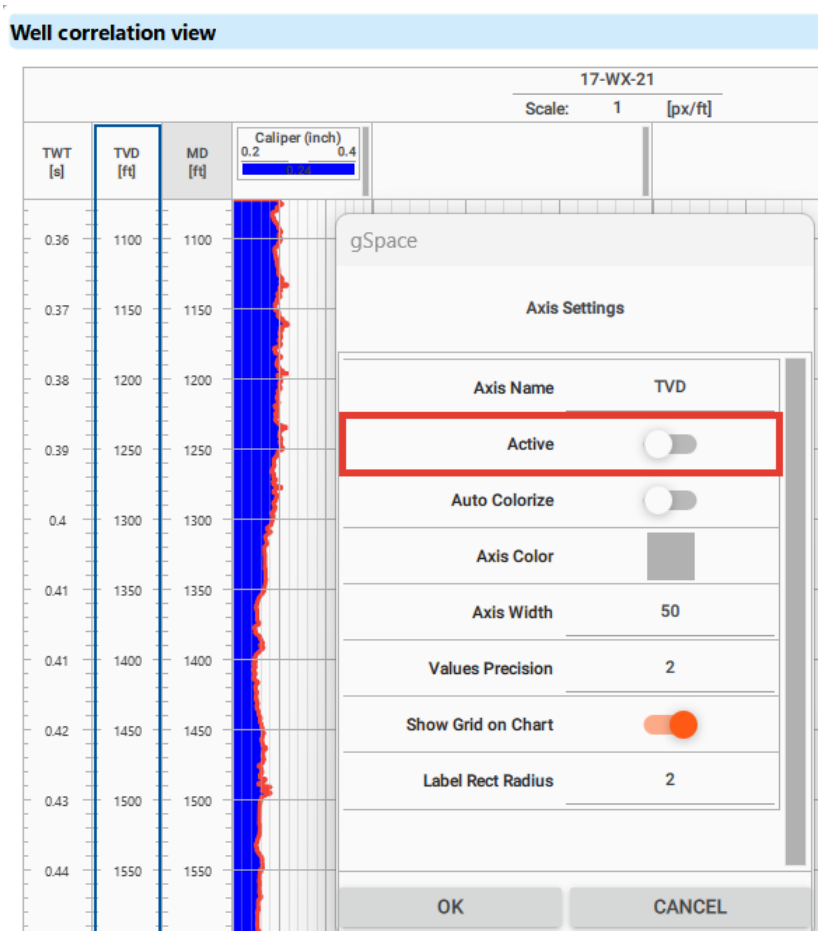
© 2026 Geomage 2003 LTD



To set a priority scale, right-click on the scale field you want to designate as the main one. This will open the settings. Enable the **Active** function, and the selected scale will become the primary one, highlighted in gray in the label field at the top.

In the picture below, the active scale is set to **MD**. For the active scale, the **Active** function does not appear in the settings, as it's already designated as the primary scale.

© 2026 Geomage 2003 LTD

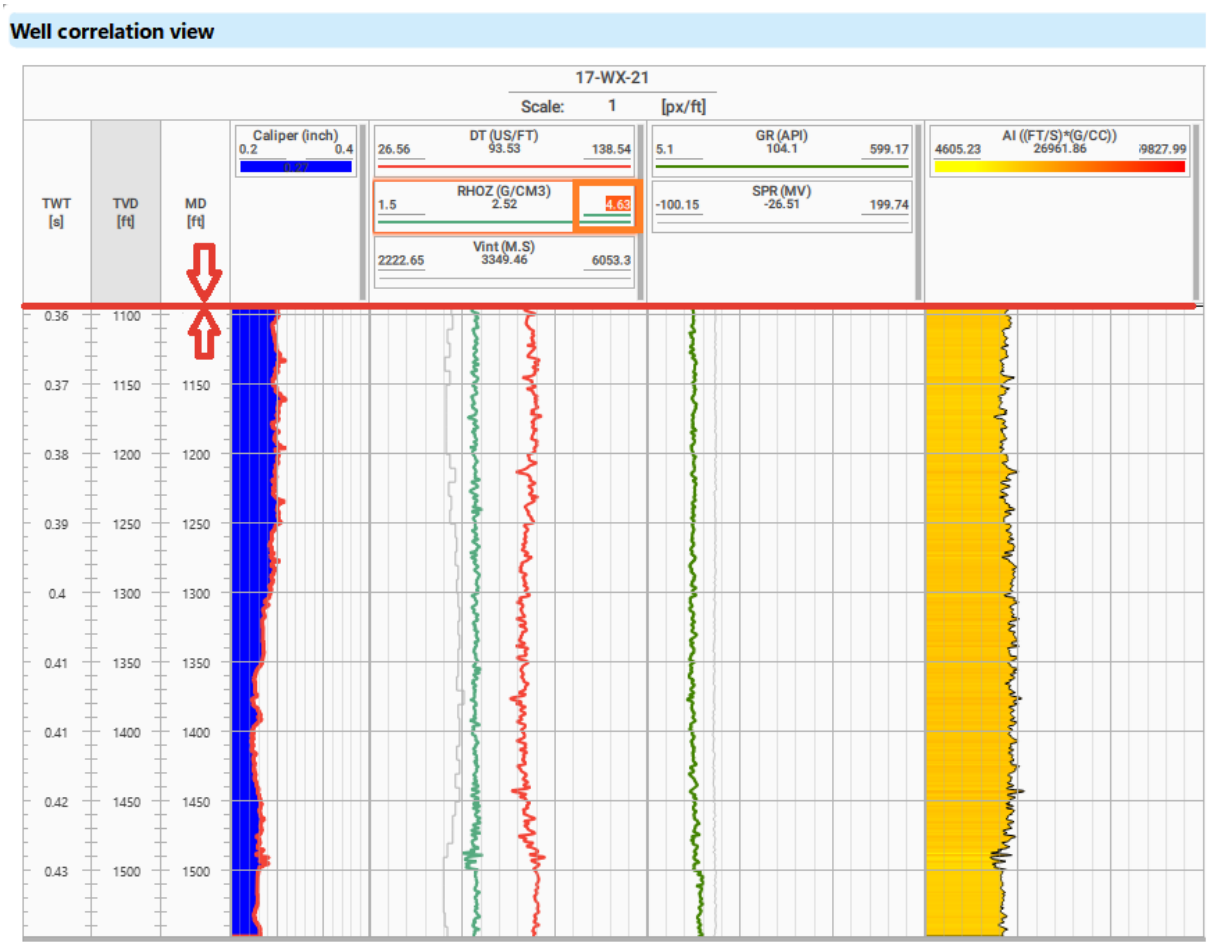


Lets add curves to the tracks as described above.

To adjust the size at the top of the panel, click and hold the left mouse button on the border (highlighted in red) of any axis field, then drag to resize the field.

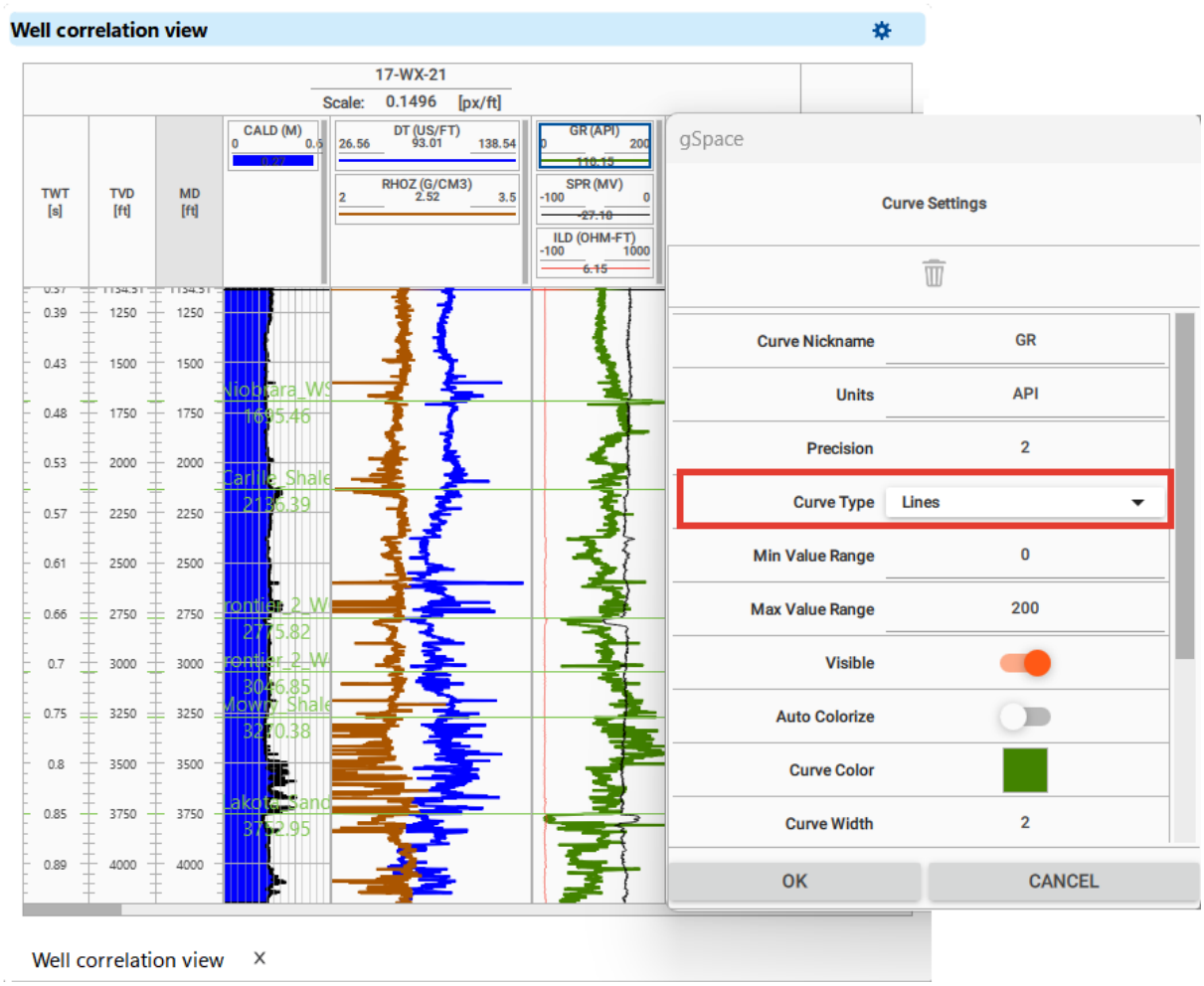
You can change the minimum and maximum values by left-clicking on the curve name field (highlighted in orange) while it highlighted.

© 2026 Geomage 2003 LTD



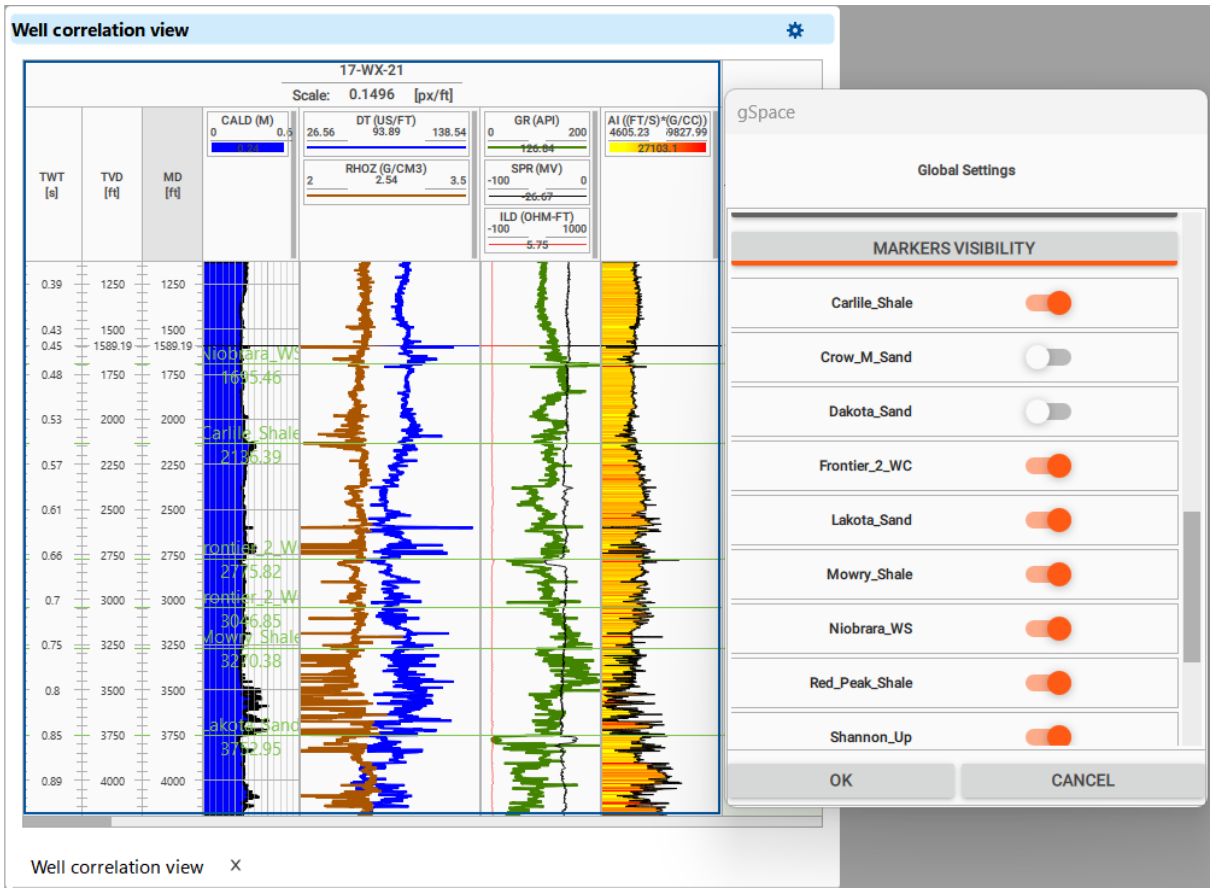
For continuous curves such as GR or Density, a convenient standard visualization type is linear.

© 2026 Geomage 2003 LTD

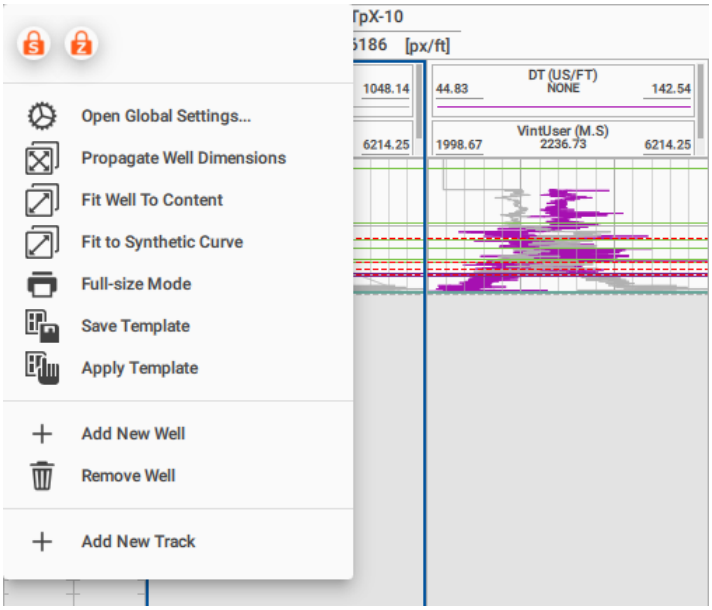


To visualize markers, open **Global Settings**, scroll down to **Markers Visibility**, and enable the markers you want to display.

© 2026 Geomage 2003 LTD



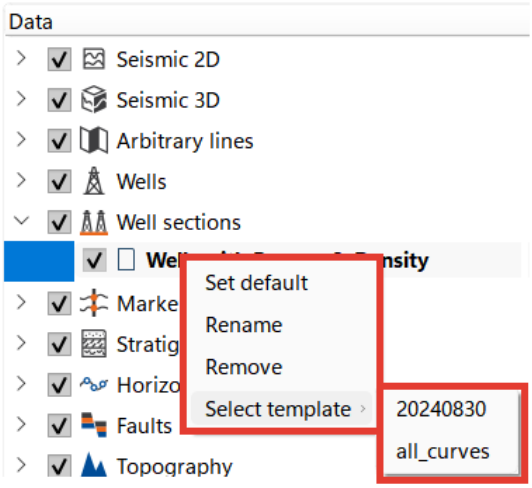
Once you have set up a Well template for correlation that suits you, save it and it will always be available in your project. To save the current setup, right-click the well name at the top of the well section and choose **Save Template**.



© 2026 Geomage 2003 LTD

To apply a saved template directly in the view, right-click the well name and choose **Apply Template**, then pick the template by name from the list that appears. This is available in every well log view, including the **Well log** view of the Wells data workspace 232. A Well correlation view 296 lists the templates saved for well sections and applies the chosen one to the current well section, while a single-well **Well log** view lists the templates saved for single-well views. If no template has been saved yet, g-Space reports that there are no saved templates to apply and asks you to use **Save Template** first.

A saved well template can also be applied from the Data Manager: right-click on the desired section within the Well Sections and choose **Select template**.



You can also manage saved templates from the Wells bar 46. In the **Templates** section, use **Select template** to apply a saved template to the active Well log or Well correlation view. Use **Manage well templates** to select a saved template and then rename or delete it. A new name must not be empty or duplicate another template name; deletion requires confirmation.

9.2.2 Working with Markers

Working with Markers in Well Correlation

1. **Creating and Managing Marker Sets:**

In the Data Manager 124, right-click **Markers** and choose **Add marker set** to create a new set. Marker sets allow you to organize markers by stratigraphic sequence, lithology, or other criteria, facilitating more effective correlation.

2. **Adding and Editing Markers:**

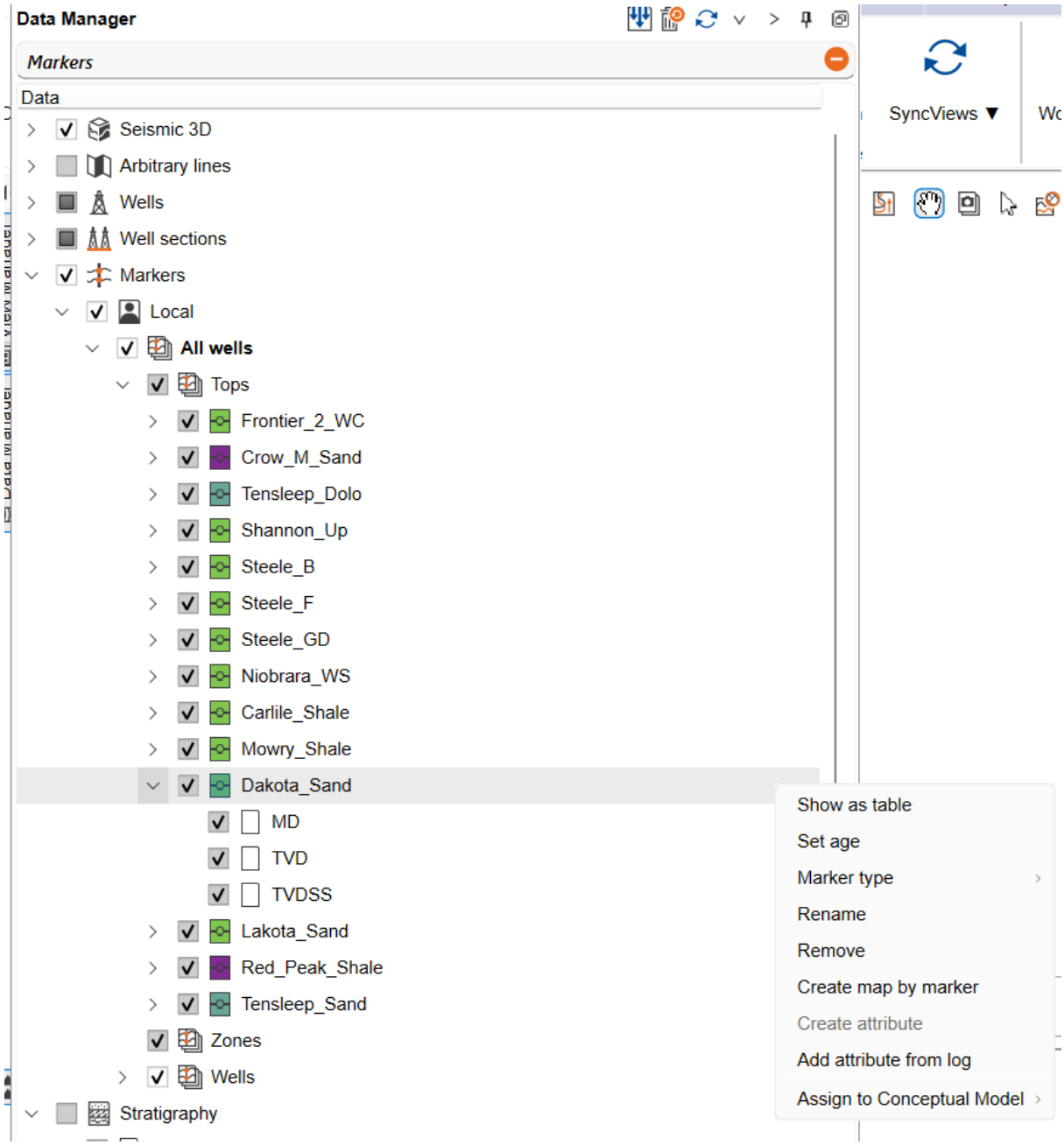
© 2026 Geomage 2003 LTD

Once a marker set is created, you can add individual markers by specifying their depth or time position.
You can also adjust marker positions, correct any inaccuracies, and delete markers as needed. Markers can be grouped to represent specific intervals or sequences, further enhancing the clarity and usefulness of your correlation.

To access marker settings in the **Data Manager**, right-click a marker under **Tops**. The menu lets you adjust its age, specify whether it represents a horizon, fault or intrusion, rename or delete it, and create a map from it. Choose **Show as table** to open that marker across every well in a table; see Markers 150 for the complete marker and zone table contents.

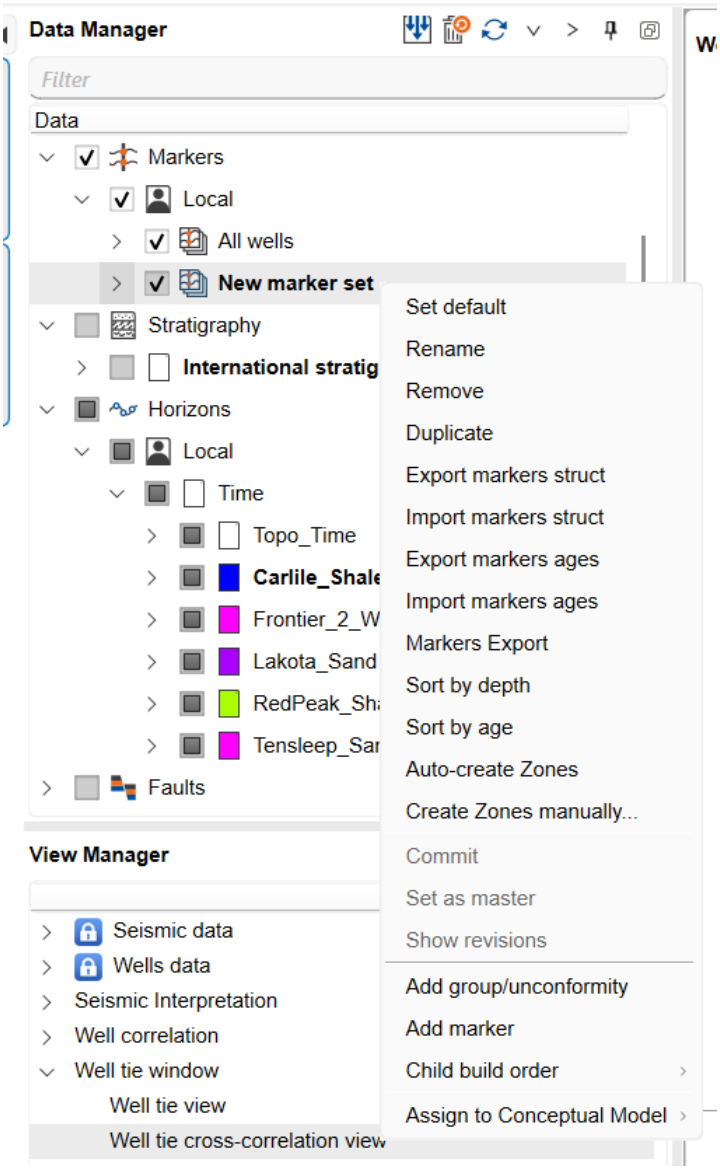
By default, each marker has the attributes **MD**, **TVD** and **TVDSS** in the **Data Manager**. These attributes are listed as sub-items under the marker in the Data Manager tree, and further custom attributes can be added next to them.

© 2026 Geomage 2003 LTD



To configure a **Marker set**, right-click on it to open an expanded menu of options. It also offers **Auto-create Zones** and **Create Zones manually...**, which build zones from the tops of the set; see Markers 150 for details.

© 2026 Geomage 2003 LTD



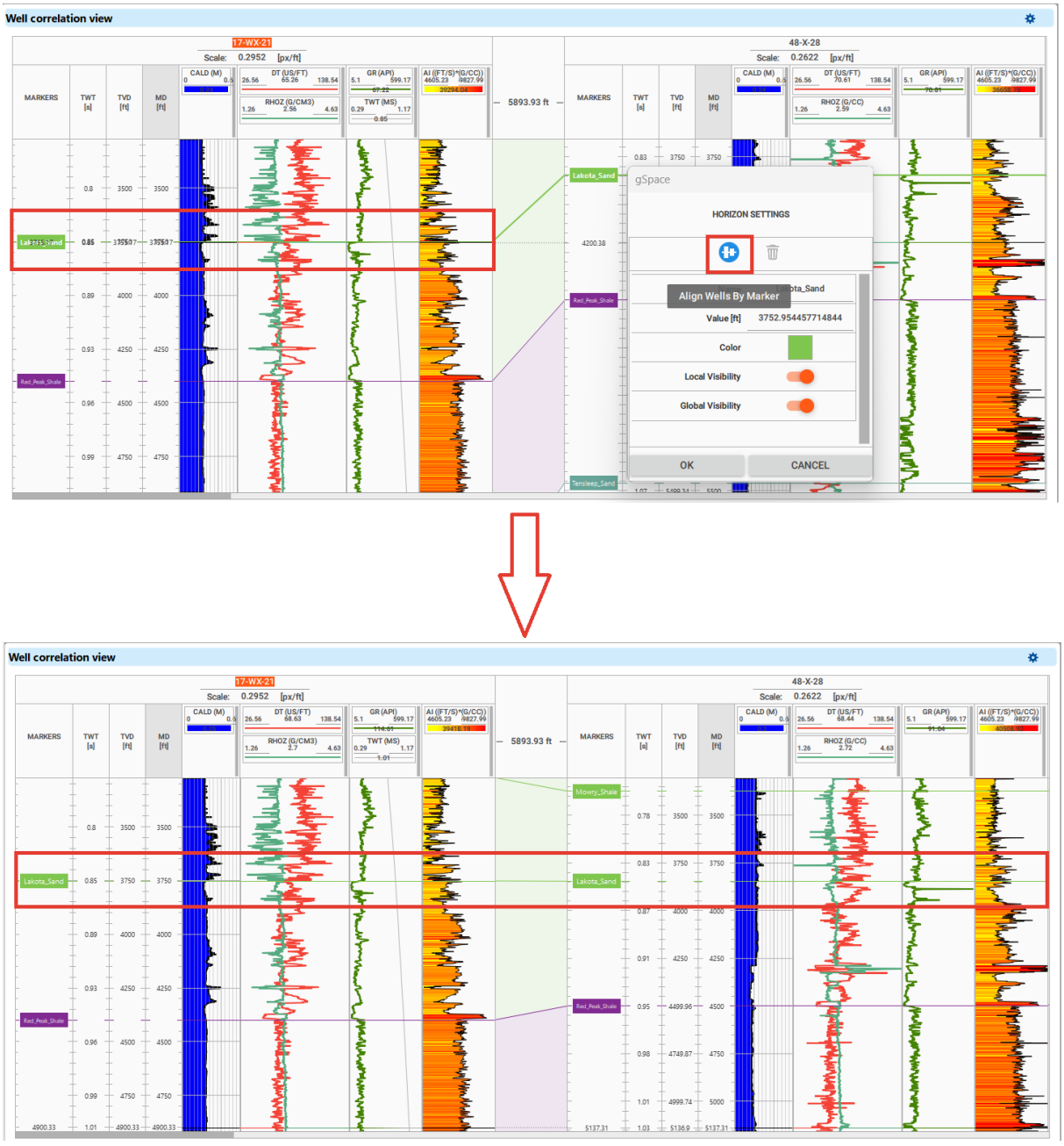
When performing correlation, it is often convenient to flatten the entire correlation scheme with a boundary.

To activate flattening, move the cursor to the desired marker. When the two-way arrow appears, the selected horizon line becomes bold, right click and open the horizon settings menu, click the first icon, highlighted in red in the picture, and the correlation diagram will align with the selected horizon. For this functionality "F" hot key is also available.

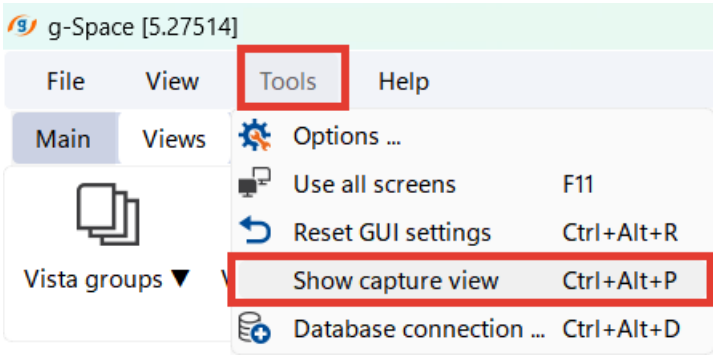
To switch between horizons, make the desired horizon active and press the "F" key again. To exit flattening mode, left-click in the axis field and press the "F" key once more. The horizons will then return to displaying their actual depth values.

To exit horizon selection mode, press the Escape key.

© 2026 Geomage 2003 LTD



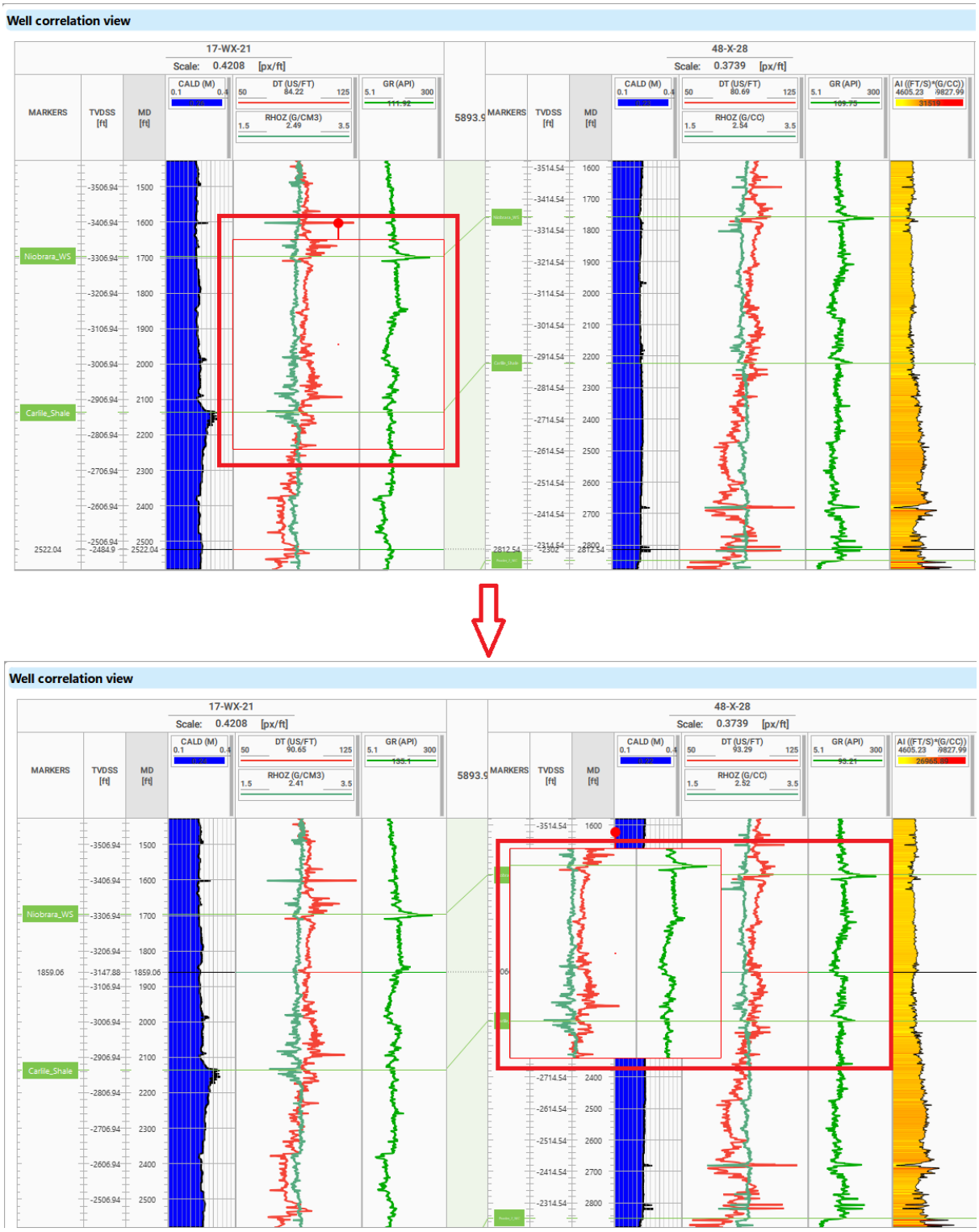
© 2026 Geomage 2003 LTD



It's often helpful to compare the curve trajectory of one well with others. since the correlation is carried out from key wells with known markers to new ones. To facilitate this, you can use the "Ghost Curves" feature.

Navigate to the **Main menu**, select **Tools**, and then choose **Show capture view**. Highlight the section of the screen you wish to compare and drag it to the desired location. To exit this mode, simply press Escape.

© 2026 Geomage 2003 LTD



During well correlation, you might need to adjust the position of markers as you refine your geological model. To add or change marker positions:

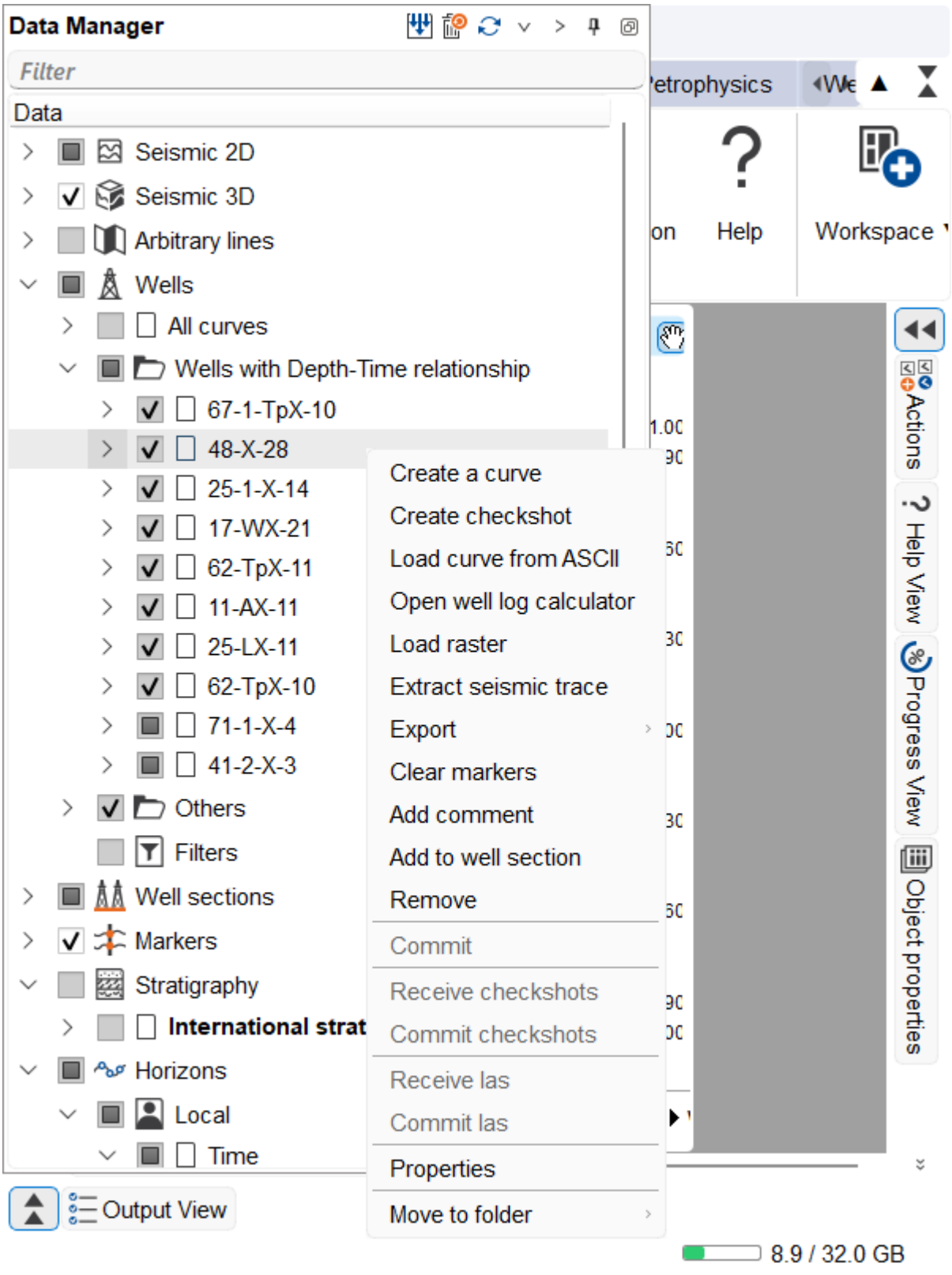
© 2026 Geomage 2003 LTD

1. **Adding a Marker:** Right-click the track of the well at the depth or time point where you want the marker, and open the **Add Markers** submenu. It lists the markers the marker set already holds; choosing one adds that marker to this well at the right-clicked depth. **Add New Marker...** at the end of the submenu opens the **ADD GLOBAL MARKER** window, where you set **Marker Name**, **Marker Color** and **Marker Position**. Confirming that window creates the marker, places it in that well and makes it the active marker at once, so no separate activation step is needed.
2. **Correlating the Active Marker:** While a marker is active, the mouse pointer shows that marker's name in the marker's own colour. Left-click the track of any other well to place the same marker at the clicked depth, so a marker is correlated across the section with one click per well; Ctrl+left-click on an existing marker line places it at that spot. Clicking an existing marker's line makes that marker the active one, creating another new marker transfers the active state to the newer marker, and Esc stops placing it.
3. **Changing Marker Position:** To adjust an existing marker, click and drag it to the correct position within the well log. When you move the cursor close to a marker line, the cursor changes to a double-headed arrow. At this point, the horizon you click on will be highlighted, and its name will be displayed. Ensure the marker accurately represents the stratigraphic horizon or boundary you are correlating.

9.2.3 Well log calculator

The well log calculator creates or updates a curve in one well by applying a mathematical expression to existing log data. In the **Data Manager**, right-click the target well and select **Open well log calculator**.

© 2026 Geomage 2003 LTD



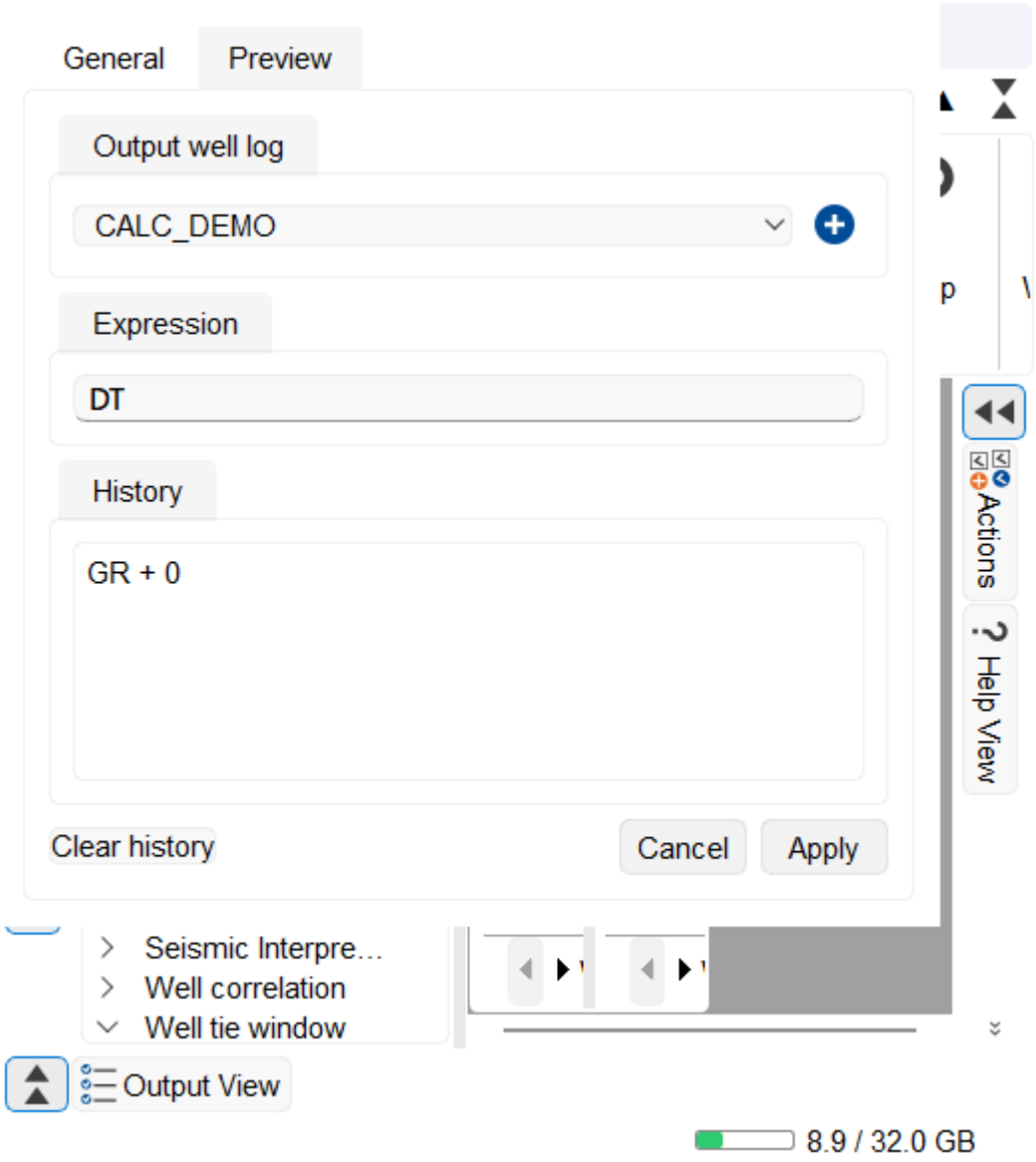
The wizard consists of two main tabs: **General** and **Preview**.

© 2026 Geomage 2003 LTD

In the **General** tab, users can define the core parameters for the calculation process. It contains the following key components:

- **Output well log:** Users can either select an existing well log curve from the dropdown menu or create a new curve by naming it. This output will store the results of the calculation.
- **Expression:** This is the central area for performing operations. Users must input an arithmetic expression using the available well log curves (e.g., porosity, density) and mathematical operators. Expressions can be as simple or complex as needed, enabling flexibility in creating derived curves. Refer to the Trace Headers Math 521 for a description of the available mathematical functions.
- **History:** g-Space automatically saves previous expressions, making it easier for users to reapply calculations they have used in the past without having to rewrite them.

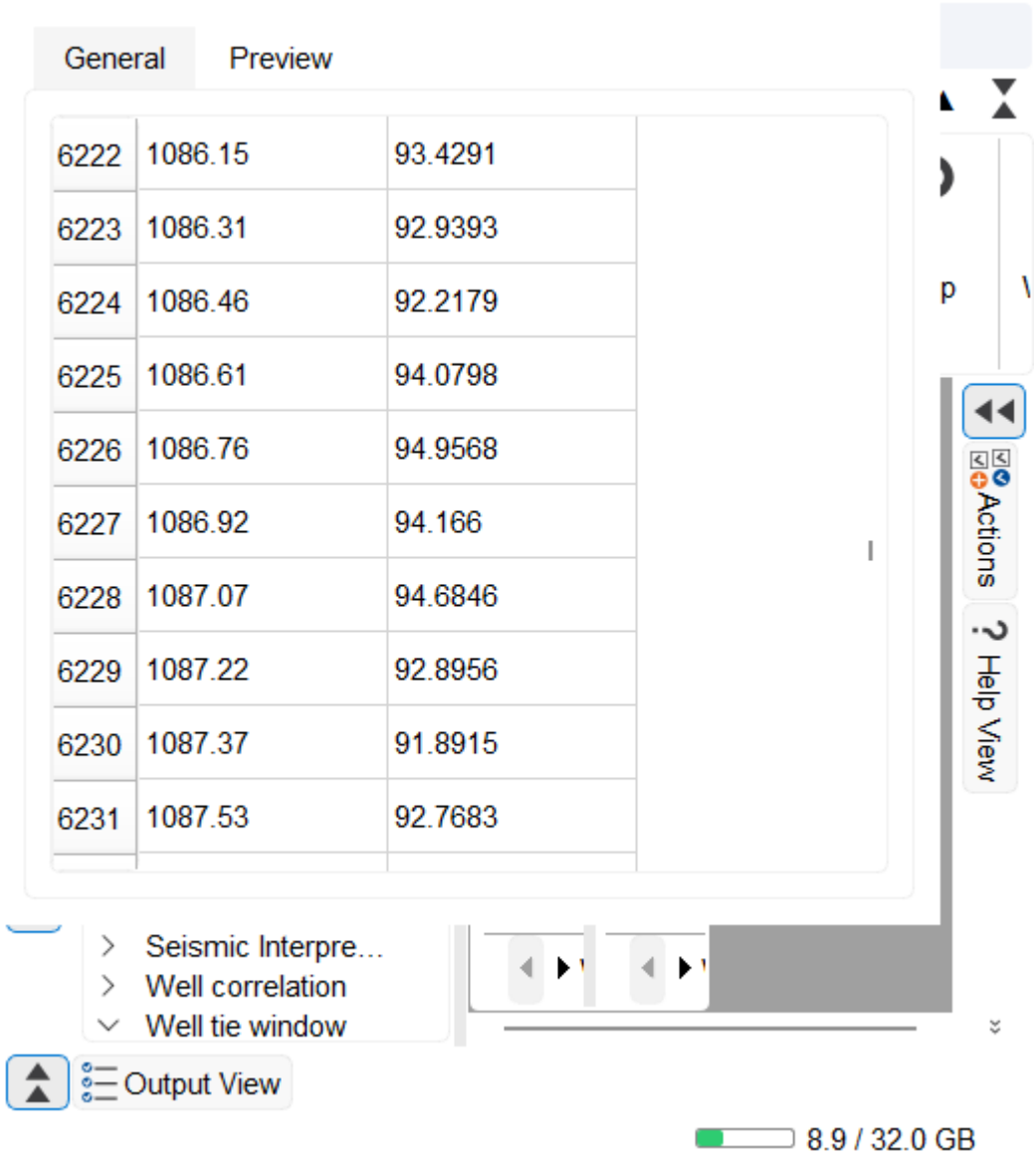
© 2026 Geomage 2003 LTD



The **Preview** tab provides a real-time visualization of the data before finalizing the calculation. It consists of two columns:

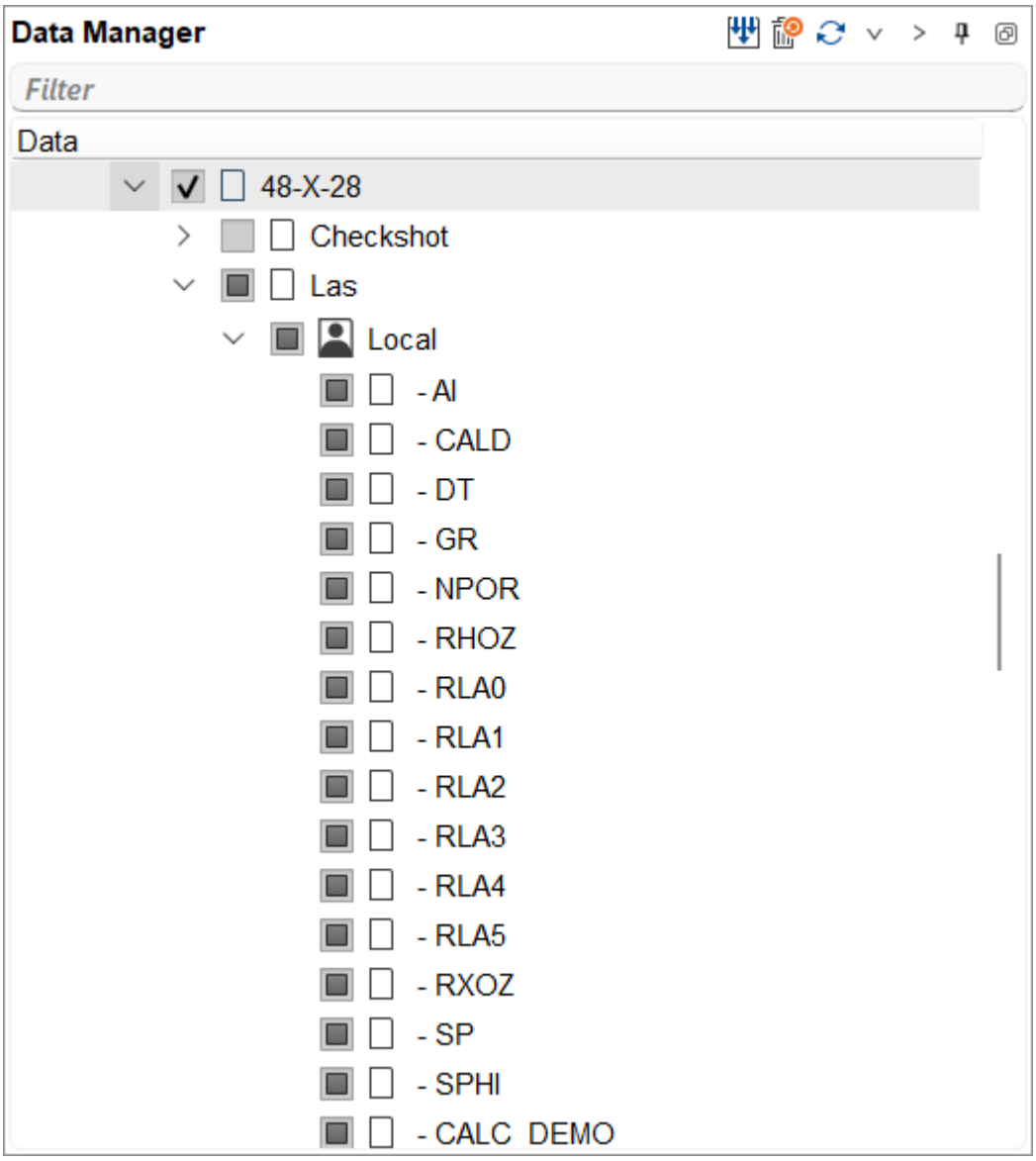
- **Depth:** Displays the depth intervals for the well data in ascending or descending order.
- **Values:** Shows the resulting values based on the arithmetic expression entered in the **General** tab. This preview ensures that users can verify the output and confirm the expression before applying it to the selected well.

© 2026 Geomage 2003 LTD



Click **Apply** to run the calculation. The dialog stays open, so you can create several curves in one session; close it with **Cancel**. Each calculation creates or updates the selected output curve in the selected well's **Las** folder in the **Data Manager**.

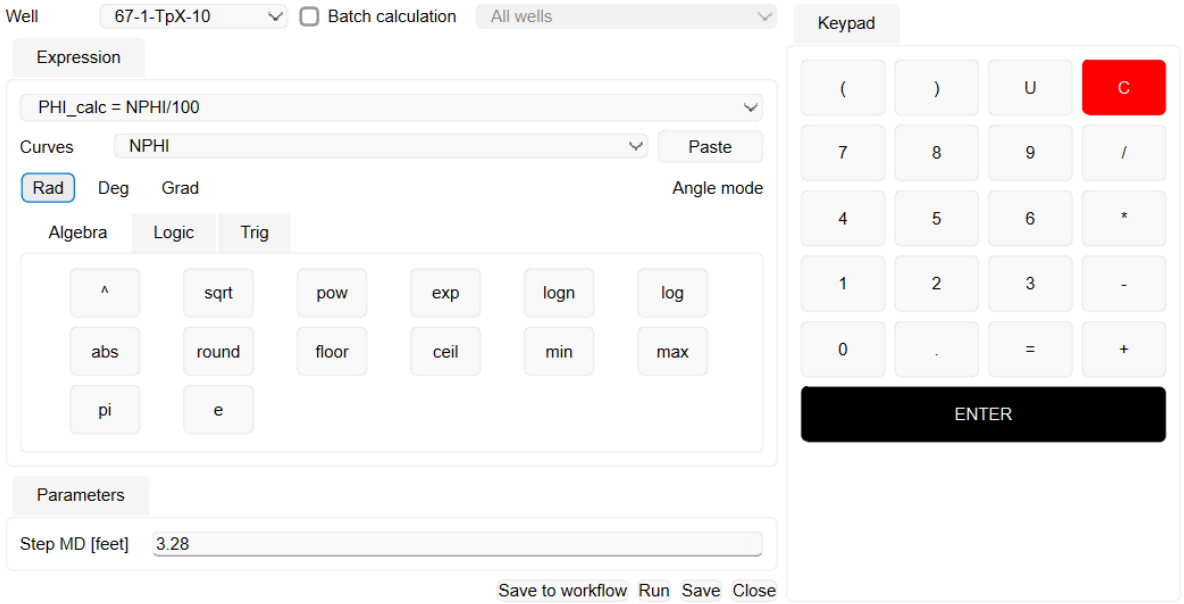
© 2026 Geomage 2003 LTD



Running the calculator from the Petrophysics ribbon

The **Log calculator** is also available from the **Petrophysics** ribbon tab, in the **Logs** group. Opened this way it provides an expanded calculator interface with a built-in keypad, function buttons, a well selector and an expression history. In this interface the output curve name and the expression are typed together in the **Expression** field using the form **name = expression** (for example, **NPHI_calc = ...**). The part before the equals sign becomes the name of the new curve.

© 2026 Geomage 2003 LTD



The main controls are:

- **Curves** and **Paste** — select an available curve (or **MD** for measured depth) and click Paste to insert its name into the expression. Use **U** for the undefined value.
- **Algebra**, **Logic** and **Trig** tabs and the keypad — insert mathematical functions, operators and digits into the expression.
- **Angle mode** (**Rad**, **Deg**, **Grad**) — sets how trigonometric functions interpret angle values.
- **Step MD** — the sampling interval, in depth, used to resample the input curves and store the result.

Use **Run**, **Save** or **ENTER** to calculate with the current settings. The dialog stays open after the calculation so you can create several curves in one session, and it does not block the rest of g-Space: while it is open you can still select another well in the **Data Manager**, scroll or zoom a log, map or seismic view, and use the ribbon. You can therefore check a curve you have just calculated in a view, then adjust the expression and run it again without reopening the calculator. This applies to the **Log calculator** only: the calculator opened from the **Data Manager**, and the **Map calculator**, **Seismic calculator**, **Marker attributes calculator** and **Grid calculator**, still have to be closed before you can work in the main window again. Only one **Log calculator** window is used at a time: while it is open, choosing **Log calculator** on the **Petrophysics** ribbon again applies the current settings to that same window instead of opening a second calculator. Use **Close** when you are finished. Each calculated curve is added to the well's LAS data in the **Data Manager**; if a curve with the same name already exists, **Run** and **Save** ask whether to replace it.

Click **Save to workflow** to add the expression, output curve, angle mode, depth step and selected well scope as a Log calculator job in the current workflow without running the calculation. g-Space confirms with *Log calculator step saved to the current workflow.*, and the

© 2026 Geomage 2003 LTD

step appears in the **Jobs** panel of the Workflows wizard as **Log calculator**. When you open a saved job from Workflows 535, its settings are restored and the dialog shows **Save** and **Cancel**; use **Save** to update the job or **Cancel** to leave it unchanged.

Executing the workflow runs the calculation with the stored settings, with three differences from a run in the open dialog:

- The inputs are checked before anything is written. When the workflow reaches the step, g-Space re-checks the stored inputs against the current project. If a saved well is no longer in the project, or a target well no longer holds one of the curves the saved expression uses, the step fails with *Log calculator input validation failed.* followed by *Missing well(s):* listing each missing well and/or *Missing input curve(s):* listing each missing curve together with the well it is missing from. No output curve is created, and execution stops at the failed step, as described in Workflows 535. A run that reaches the calculation but produces no calculated well at all also fails, and reports the run report as its message.
- The saved well scope is resolved again at run time. With **Batch calculation** cleared, the step targets the single well that was selected when it was saved. With the **All wells** scope, the well list is resolved again on every run, so wells added to the project after the step was saved are included. With a named scope — a well filter, a **Data Manager** well folder or a well section — the step runs on the exact wells that scope resolved to when it was saved: wells that would match it now are not picked up, and any of those saved wells that have since been removed are reported by the check above.
- An existing curve is replaced without asking. A workflow run has no user present, so it overwrites a curve that already carries the output name instead of showing the confirmation.

Calculating for several wells

The well selector row at the top of the dialog controls which wells the expression is applied to:

- **Well** — choose a single well. By default the calculation is applied to this well only.
- **Batch calculation** — turn on this check box (off by default) to run the calculation on a group of wells instead of a single well. The single-well list is then disabled and the scope drop-down beside it becomes active.
- Scope drop-down — choose **All wells** to process every well in the project, or select a well filter, a Data Manager well folder, or a well section to limit the calculation to that group.

When the run finishes, g-Space reports how many wells were processed and lists any wells that were skipped, for example because an input curve was missing, the expression produced no defined values, or the output curve already existed and was not overwritten.

Expression history

© 2026 Geomage 2003 LTD

The **Expression** field is a drop-down that remembers the last five expressions you applied, so you can reapply a previous calculation without retyping it. The most recent expression appears at the top of the list, and duplicate entries are not repeated — select an entry to reuse it.

The history is stored separately for each project, and each calculator keeps its own list, so the Log calculator, Map calculator, Seismic calculator and Marker attributes calculator do not share expressions.

9.3 Lithology Interpretation

Lithology interpretation is the petrophysical step that turns raw well logs into a single lithology curve: each sample in the selected interval is assigned a lithotype code (for example sand, shale or limestone) by comparing the input curves against user-defined value ranges. The lithology curve is the basis for net-pay definition, facies modeling and property population, so it is best run early in the well-analysis workflow, immediately after the logs have been loaded and cleaned.

Launching the tool

Open the Petrophysics 49 ribbon bar and click **Lithology** in the **Lithology** group. Select the target well, or enable **Batch calculation** to process several wells or a well group from the Data Manager 124 in one run. The full parameter reference is documented in the Lithology 77 tool topic.

Recommended workflow

1. **Prepare the logs first.** Confirm the input curves are loaded, correctly named and free of null spikes. Repair problem curves in LAS editing 320 and derive any missing inputs (for example shale volume) before interpreting lithology.
2. **Choose discriminating curves.** Pick the logs that best separate the expected lithotypes (typically gamma ray, density, neutron and resistivity) and set a minimum and/or maximum threshold for each.
3. **Define the lithotypes.** Set the number of lithotypes, name each one from the lithology map and enter its curve thresholds and match rule.
4. **Restrict the interval.** Use **MD range**, **TVD range**, **TVDSS range**, **Markers range** or **Zones range** to limit the interpretation. With Zones range active, select one or more zones; the existing lithotype rules are applied only inside valid selected intervals and values outside them remain undefined.

© 2026 Geomage 2003 LTD

5. **Run and QC.** Run the calculation, then review the output lithology curve against the source logs and markers in the well log view. Adjust thresholds and re-run until the result is geologically consistent.

Recommendations

- Calibrate thresholds against a key well with core or reliable cuttings before running in batch mode across the field.
- Keep the number of lithotypes to the minimum that the logs can actually resolve; too many classes produce a noisy, unreliable curve.
- Re-run lithology interpretation whenever the underlying curves are edited or new wells are imported so the lithology curve stays consistent with the log data.

See Also

- Lithology 77
- LAS editing 320
- Shale Volume 58
- Well log calculator 558
- Petrophysics 49

9.4 Well-Tie

Well Tie view in g-Space is a powerful tool that allows you to visualize and analyze well data, generate synthetic seismograms, and perform seismic-to-well tie operations.

Preparing for Well-to-Seismic Tie

To perform a seismic-to-well tie, you must have the following data loaded:

- **Seismic Data:** 2D lines or 3D volumes.
- **Well Information:** This includes check shots, sonic logs, density logs, and optionally, impedance data.

Subscribe to our channel and watch the video on how to perform Well tie in g-Space on YouTube

In this chapter, we will guide you through the well-to-seismic tie process using the Demo project for the Teapot oil field.

We will focus on the well 67-1-TpX-10, which is equipped with sonic and density logs, as well as checkshot data. The goal is to tie this well to a 3D seismic volume by generating a synthetic

© 2026 Geomage 2003 LTD

seismogram, comparing it to the seismic composite trace at the well location, and then analyzing the outcomes.

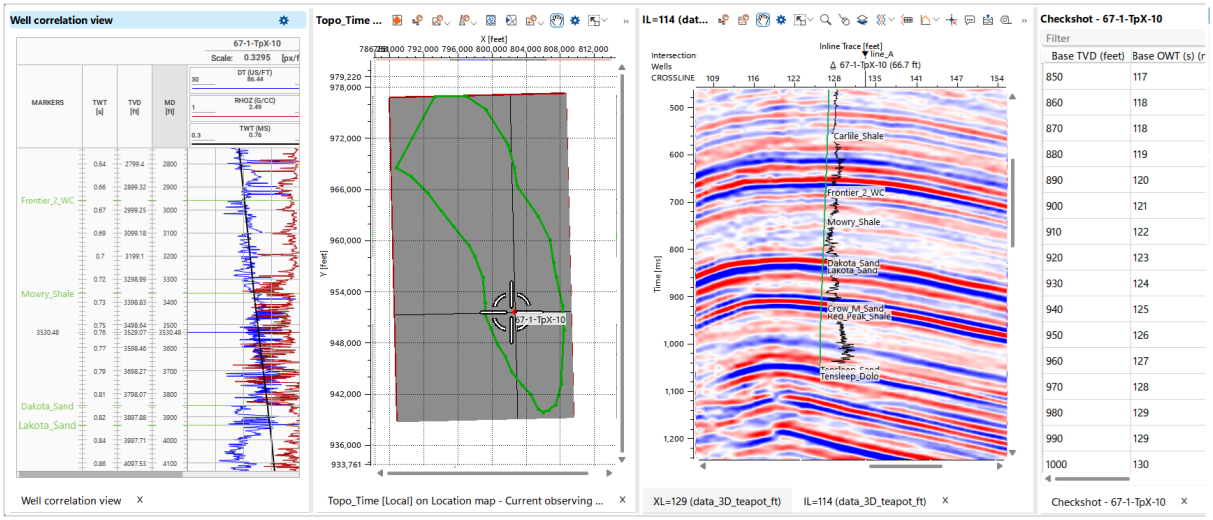
Let's have a look at the data

Well Correlation View (left panel) is used to visualize the well logs, including the sonic and density logs for well 67-1-TpX-10. The logs are plotted against depth and time. Markers are annotated alongside the logs, aiding in the correlation with seismic data.

Location Map displays the position of well 67-1-TpX-10 within the seismic survey area.

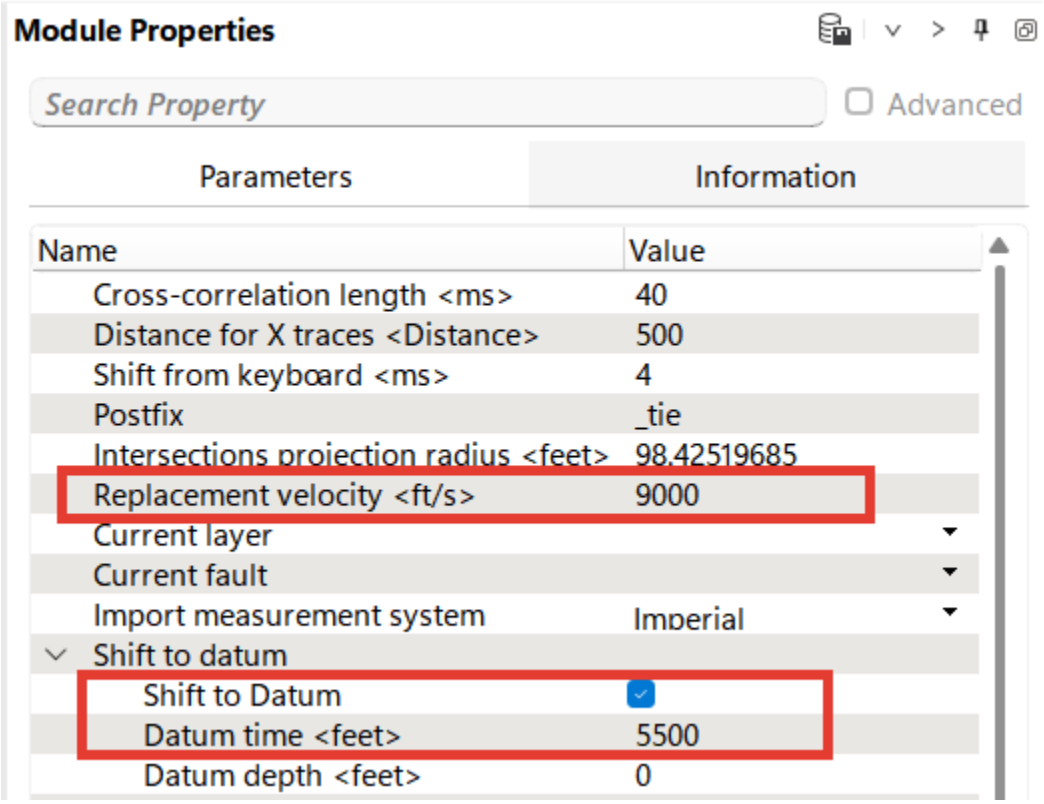
Inline Seismic View shows a vertical slice of the 3D seismic volume passing through the well location.

Checkshot Table View (right panel) provides a checkshot data for well 67-1-TpX-10, which includes both True Vertical Depth (TVD) and One-Way Travel Time (OWT). Well correlation view automatically recalculates and displays TWT values along its axis.

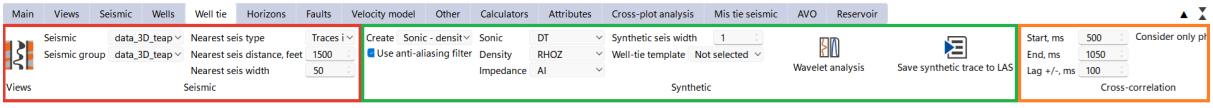


Please verify the 3D seismic reference level and replacement velocity in the Module Properties. Ensure that the **Datum** is set to 5500 feet and the **Replacement Velocity** is set to 9000 ft/s.

© 2026 Geomage 2003 LTD



Navigate to the Well Tie Bar, which will be the workspace for this process.

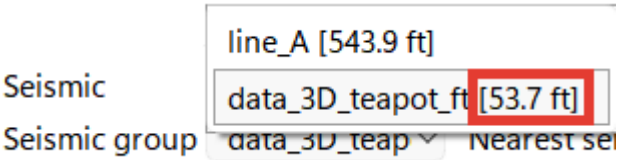


Setting Up the Well Tie Bar

The **Well Tie Bar** is divided into three main sections: **Seismic**, **Synthetic**, and **Cross-correlation**.

Seismic section (highlighted in red) allows the user to select the seismic lines or volumes that are near the selected well. It displays options to choose specific seismic data from the drop-down menu, such as the seismic volume or group. In this Demo project, we will be performing the well tie to a 3D seismic volume.

The **Nearest Seis Distance** field displays the distance from the selected well to the nearest seismic line or volume. For example, in this case, the distance is 53.7 feet.

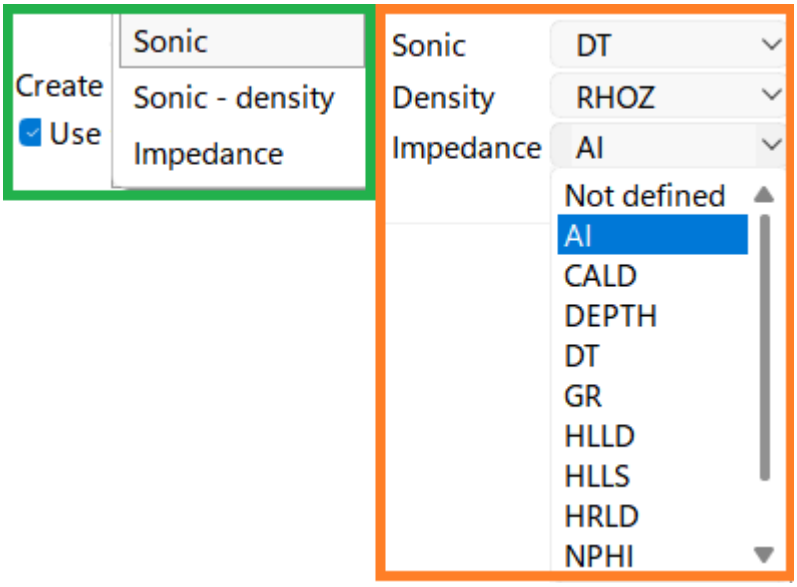


© 2026 Geomage 2003 LTD

The user can define a specific distance for the nearest seismic data using the **Nearest Seis Distance** setting. For instance, if a distance of 1,500 feet is specified, only seismic lines within this radius from the well will be considered.

Nearest Seis Width defines the number of seismic traces to be displayed around the well location. In this case, we set it to 50 traces.

Synthetic Section (highlighted in green) is dedicated to generating and analyzing the synthetic seismogram. It includes tools for creating synthetic traces based on the sonic and density logs and performing wavelet analysis.



To create the synthetic curve, you need to specify the type of data that will be used in the process. In our scenario, we have both **acoustic (sonic)** and **density logs** available. To proceed, in the drop-down menus (highlighted in orange), select the appropriate curves that correspond to the available data:

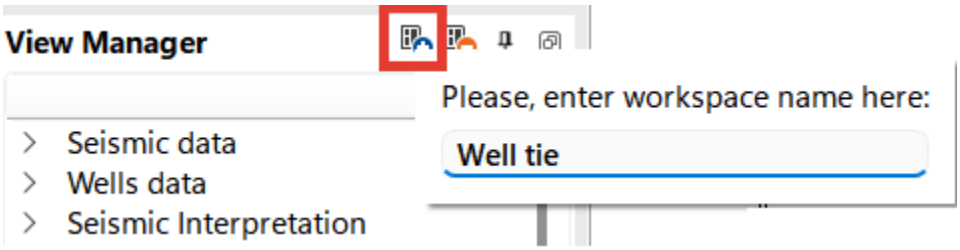
- For the **Sonic** log, choose the curve labeled **DT**
- For the **Density** log, select the curve labeled **RHOZ**

Use anti aliasing filter we will set by default, checked.

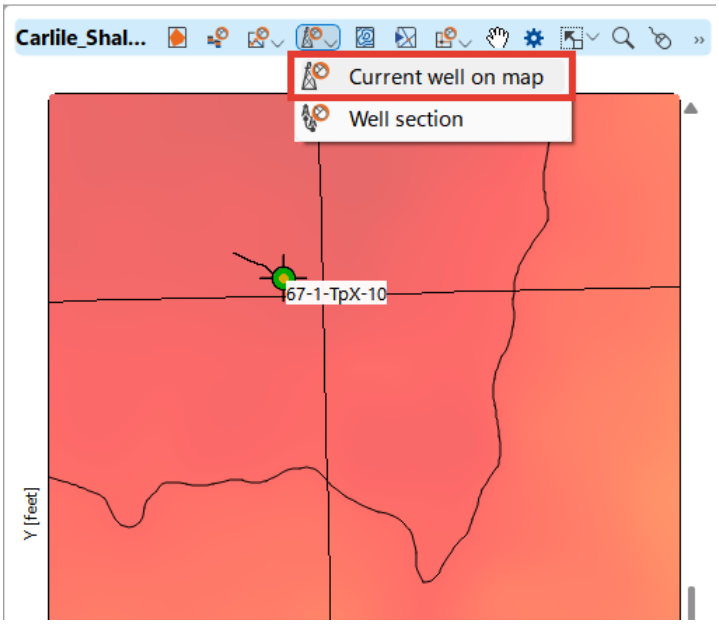
Synthetic Seis Width allows you to define the width of the synthetic seismogram by specifying the total number of seismic traces to include. For our project, we will set this value to **1 trace** to generate a single synthetic trace.

© 2026 Geomage 2003 LTD

To visualize the well-to-seismic tie process, start by creating a new workspace in the **View Manager**.



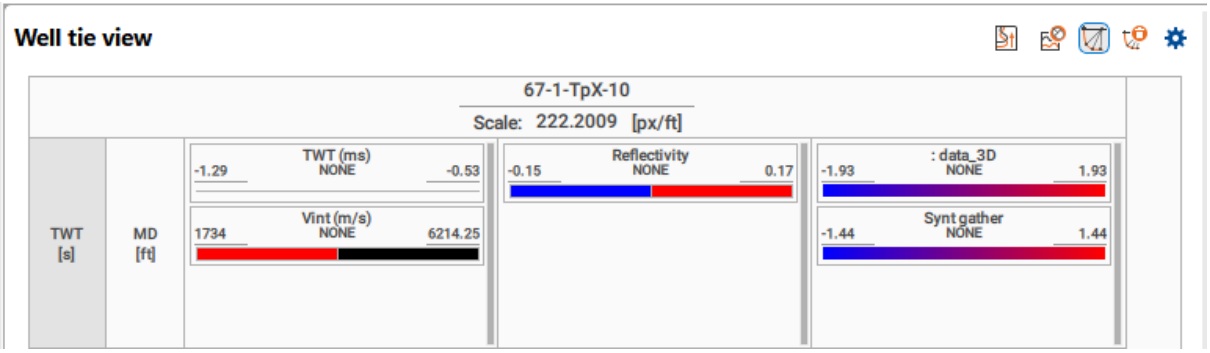
To display the tie, you'll need to include a **Location Map** where the wells will be displayed. As you choose a well with time-depth data, it will be visualized at Well tie View.



Following this, create a **Well Tie View** by clicking from the Well tie bar. This function is also available in Views Bar 38, choose the same button Well tie 287.

The default **Well Tie View** will open in front of you. Let's review and configure it.

© 2026 Geomage 2003 LTD



The display consists of three tracks showing the following data:

- Two Way Time
- Interval velocities
- Synthetic trace
- A seismic section with the synthetic trace overlaid at the well intersection

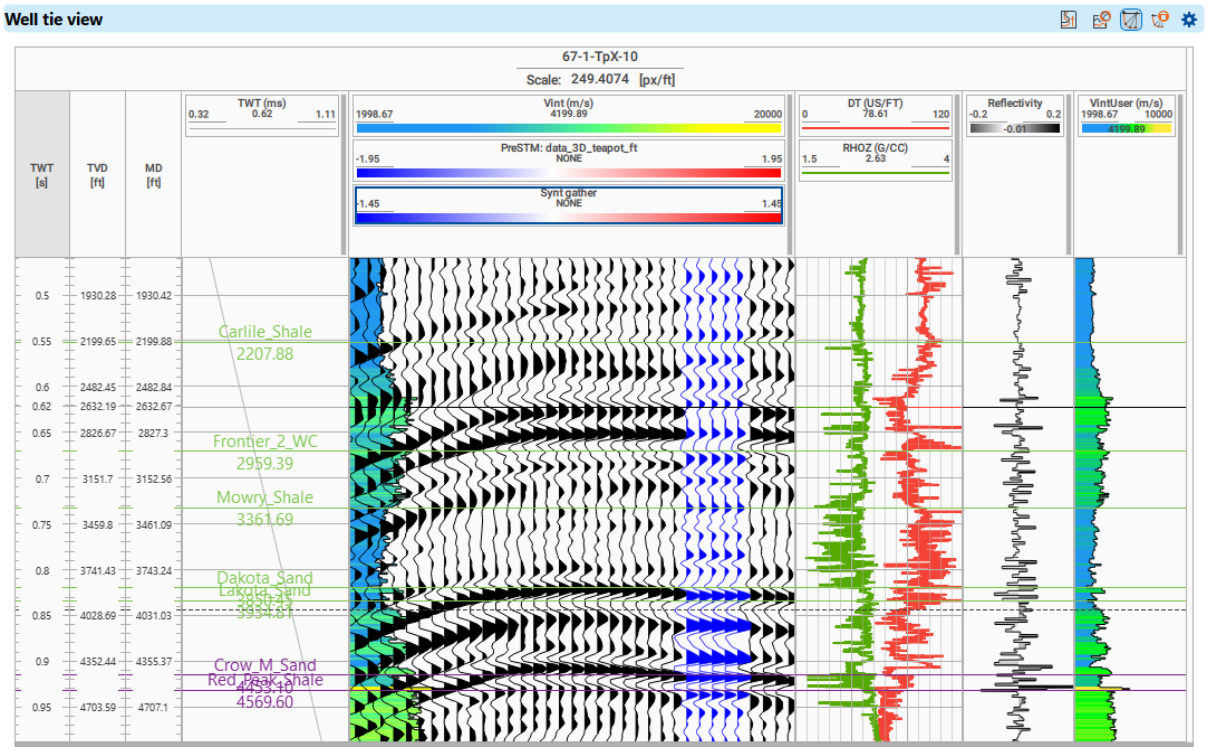
You can also check all the settings in the Module Properties 197 panel.

Let's configure the template and adjust it to a view that suits your preferences. The full settings description for **Well Tie View** are available here 287. The curve settings are similar to the settings in Well correlation 538

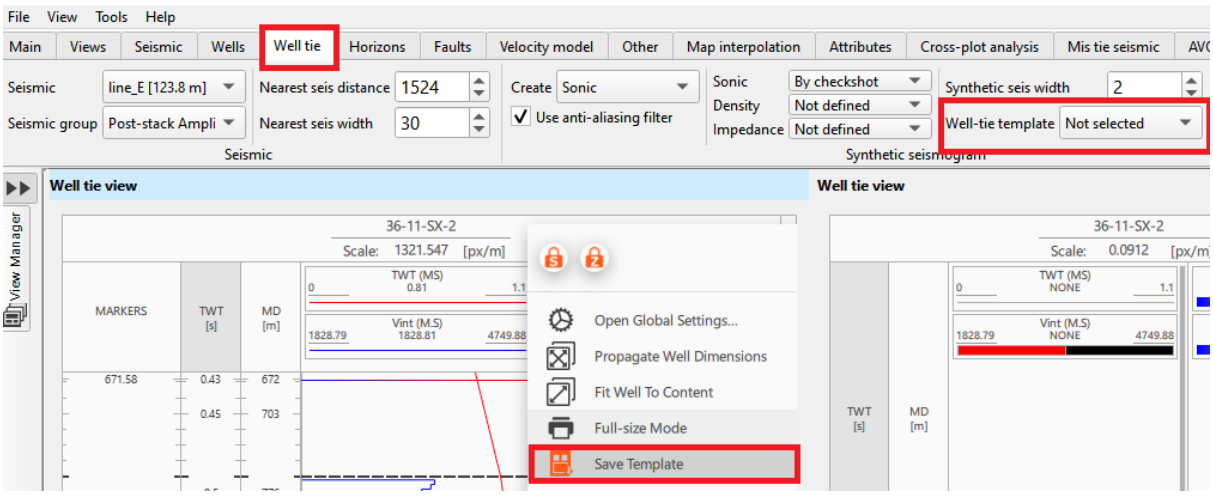
By right-clicking, you can access global settings, where you can easily add scales, such as TVD, which is especially useful for horizontal wells.

For this example, the seismic section was configured to display as wiggles, and the number of synthetic traces was adjusted for easy visual comparison. Markers were also visualized along with the well logs to enhance the interpretation.

© 2026 Geomage 2003 LTD

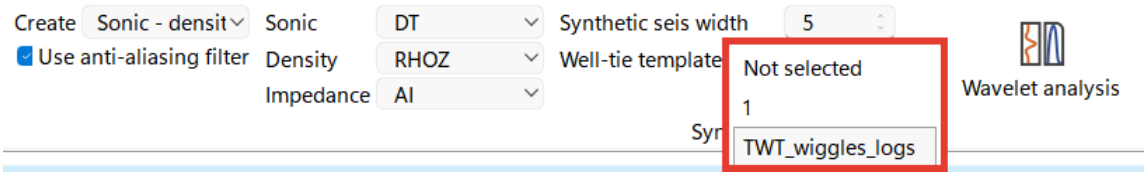


To save the settings as a template, right-click in the well name field and select **Save Template**.



To apply the template, navigate to the **Well Tie Bar** and select the desired template from the drop-down menu.

© 2026 Geomage 2003 LTD



The **Wavelet analysis** option lets you select an existing wavelet or create one for the synthetic seismogram.

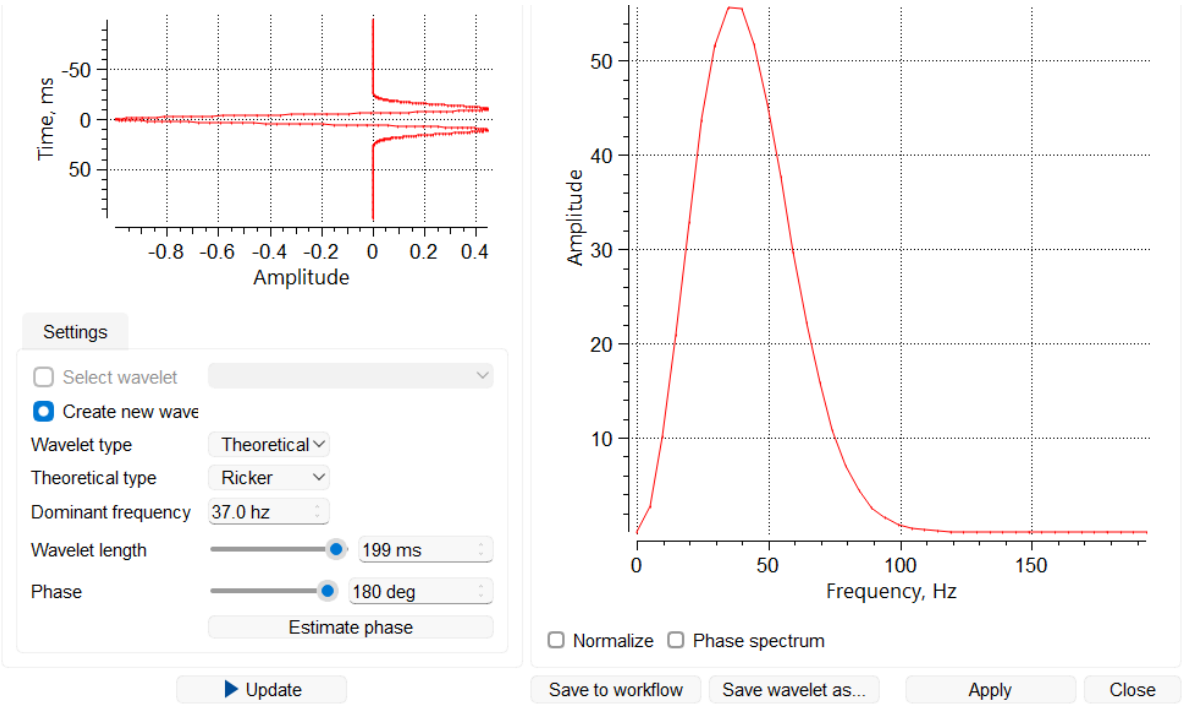
The **Wavelet analysis** window is not modal, so you can keep working in g-Space while it is open. It is divided into a **Wavelet** side, where you choose the input and set the parameters, and a **Spectrum** side, which plots the spectrum of the current wavelet.

When the project contains seismic data, the **Seismic data** group at the top of the window lists the available datasets. Type in the search box above the list to filter it by name, then select the dataset the wavelet is estimated or extracted from.

By default, the system uses a Ricker wavelet. Select **Create new wavelet**, then choose one of the following **Wavelet type** options:

1. **Theoretical:** Creates a Ricker, Gaussian, or Ormsby wavelet from the specified parameters.
2. **Statistical:** Estimates a wavelet from the selected seismic data.
3. **Extracted:** Extracts a wavelet from seismic traces at a chosen location.
4. **From well log:** Derives a wavelet from the current well's sonic and density logs and the seismic trace at the well location.

© 2026 Geomage 2003 LTD



When **Wavelet type** is set to **Statistical**, the **Data range** group defines which seismic traces the wavelet is estimated from. The start and end fields accept only values within the extent of the dataset selected under **Seismic data**, and are pre-filled with its full extent.

Leave **Limit data range** cleared to estimate the wavelet from the seismic trace at the current well location. Select it to enable the range fields and estimate the wavelet from every trace inside the range instead:

- For a 3D volume, set **Inline start**, **Inline end**, **Xline start** and **Xline end**.
- For a 2D line, set **CDP start** and **CDP end**; the Xline fields do not apply and are hidden.

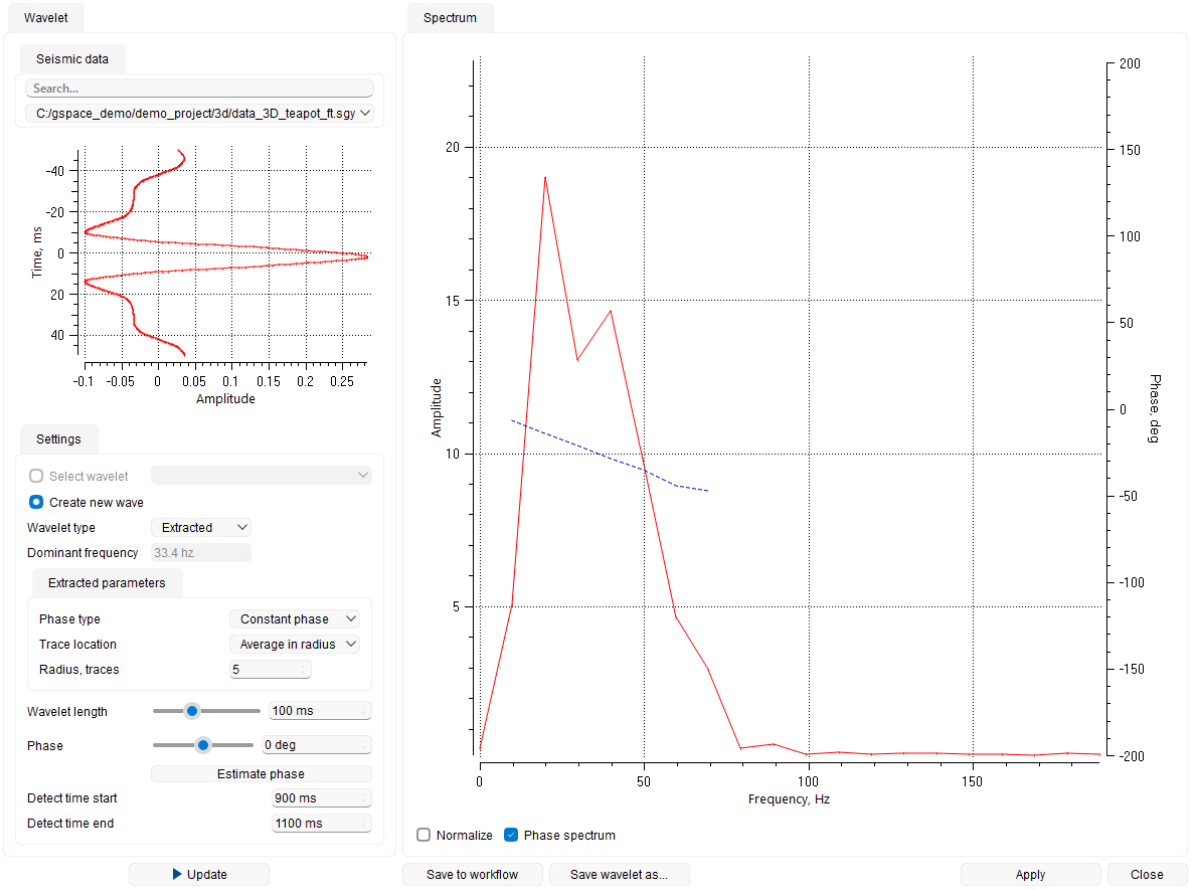
When **Wavelet type** is set to **Extracted**, the **Extracted parameters** group defines how the wavelet is taken from the selected seismic data:

- **Phase type**: select **Constant phase** to give the whole wavelet a single estimated phase, or **Variable phase** to keep the phase as extracted from the data.
- **Trace location**: **Wellhead** uses the trace nearest to the well, **Average in radius** averages the surrounding traces within the number of traces set in **Radius, traces**, **Composite trace** uses the composite trace along the wellbore, and **Manual** uses the trace at the **Inline** and **Crossline** you enter. The additional fields appear only for the location that needs them.

Set the length of the wavelet with the **Wavelet length** slider or its value box, and rotate its phase with the **Phase** slider or value box; click **Estimate phase** to estimate the phase from the data. For the **Statistical**, **Extracted** and **From well log** types, **Detect time start** and **Detect time end** limit the time interval the wavelet is estimated from.

© 2026 Geomage 2003 LTD

The **Spectrum** side plots the amplitude spectrum of the current wavelet. Select **Normalize** to normalize the amplitudes and **Phase spectrum** to add the phase curve with its own **Phase, deg** axis on the right. Use the mouse wheel over the plot to change its scale.



Additionally, users can save the synthetic trace to a LAS file for further analysis.

After changing parameters, click **Update** to recalculate the wavelet preview. Click **Apply** to use the current wavelet in the well tie and refresh the synthetic trace, **Save wavelet as...** to save the wavelet with a name, or **Close** to dismiss the window.
Click **Save to workflow** to add the selected input and parameters as a **Wavelet analysis** job in the current workflow 535 without running it immediately. The window remains open; the wavelet is created when the workflow runs.
To edit a saved job, open it from the **Workflows** window. The window restores the saved settings and shows **Save** and **Cancel**; click **Save** to update the job or **Cancel** to leave it unchanged.

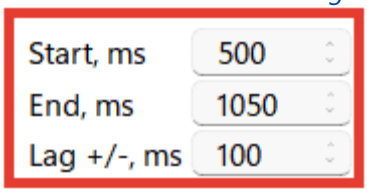
Cross-correlation section handles the cross-correlation process between the synthetic seismogram and the seismic data. It includes settings for the time range (start and end times)

© 2026 Geomage 2003 LTD

and the lag in the Well tie bar. The user can adjust these settings to optimize the match between the synthetic and seismic data.

Note

For accurate results, it is recommended to set the cross-correlation calculation window to at least 2-3 wavelet lengths.



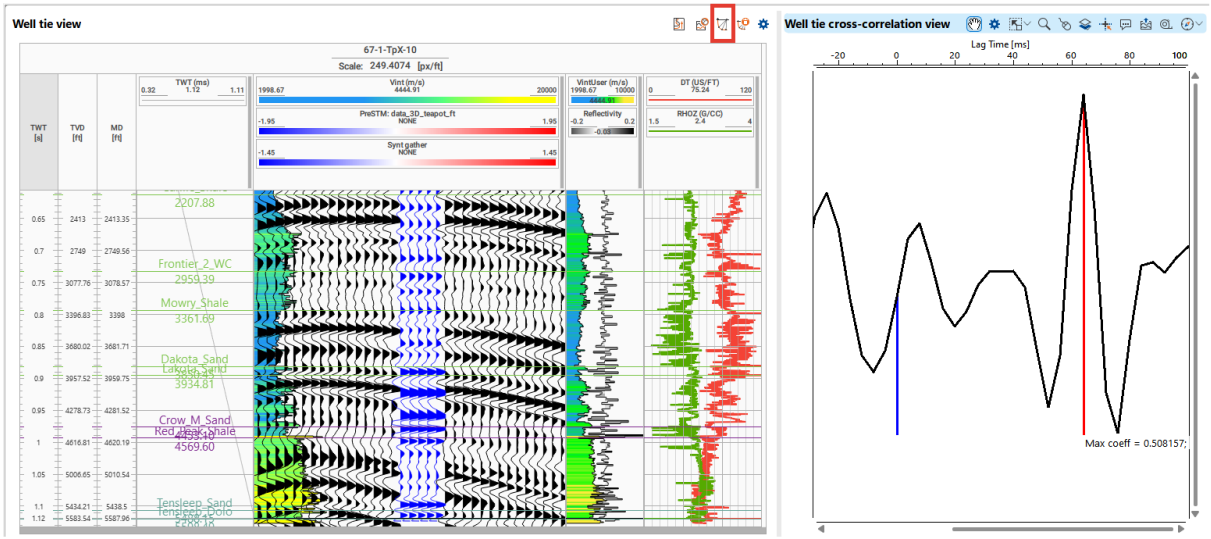
Let's move on to Seismic to Synthetic Comparison and Time-Depth Adjustment

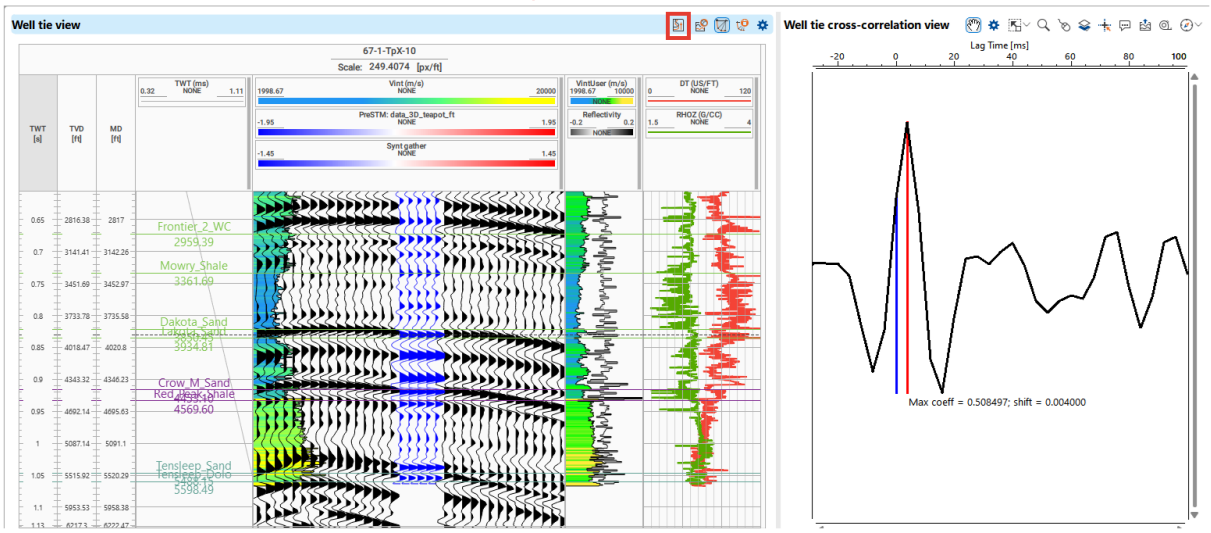
Begin by identifying and matching known reference reflections between the seismic and synthetic data.

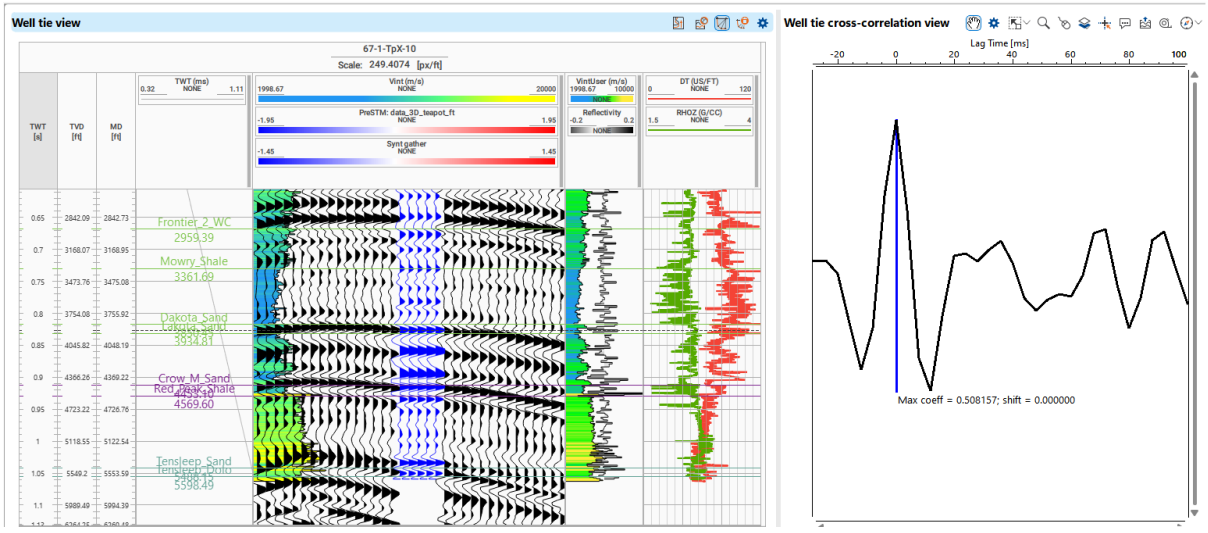
Activate the anchor change function. Then, select a reference reflection and double-click the left mouse button to place an anchor on it. Anchors act as tie points, allowing users to align the seismic data with the synthetic trace. Hold the left mouse button to drag and position the anchor at the desired interval.

For optimizing the correlation, use the auto-shift function, which automatically adjusts to achieve the maximum cross-correlation coefficient.

© 2026 Geomage 2003 LTD

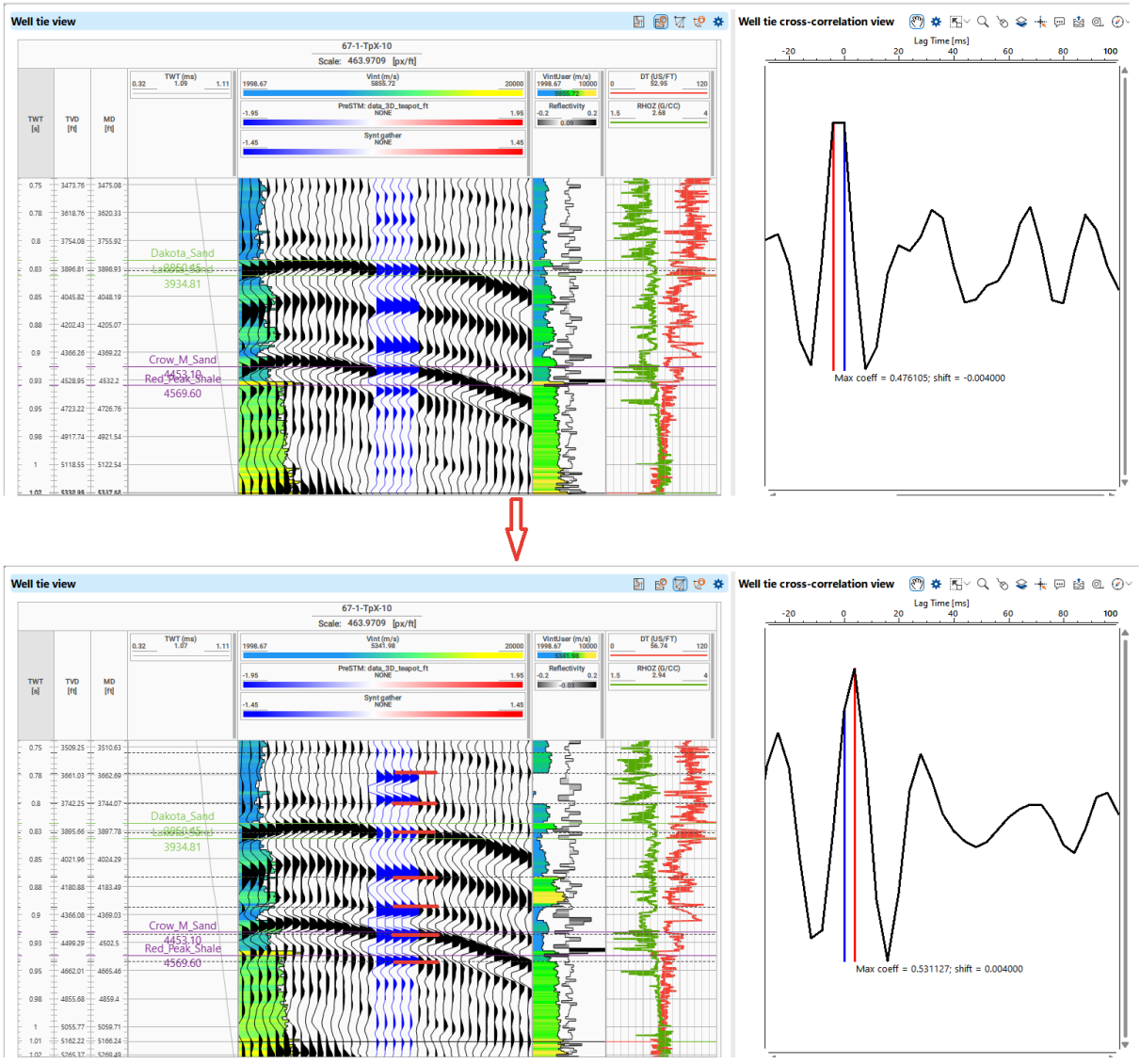






© 2026 Geomage 2003 LTD

If you are working on a specific interval, we recommend reducing the correlation interval and placing a series of anchors over a larger scale. Visually inspect the alignment of key reflectors between the synthetic and the seismic trace. And correlate them by using anchors (highlighted in red on picture below).



© 2026 Geomage 2003 LTD

Note that as the curve changes, you see changes in the cross-correlation graph and in the time-depth table. There are two new columns for Result TVD and result OWT.

Checkshot - 67-1-TpX-10

Filter

Base TVD (feet)	Base OWT (s) (ms)	Result TVD (feet)	Result OWT (s) (ms)
849.990043	116.710000	849.990043	118.710000
859.989957	117.600000	859.989957	119.600000
869.989970	118.495000	869.989970	120.495000
879.989984	119.415000	879.989984	121.415000
889.989997	120.380000	889.989997	122.380000
899.990013	121.305000	899.990013	123.305000
909.990026	122.190000	909.990026	124.190000
919.990039	123.085000	919.990039	125.085000
929.989954	123.980000	929.989954	125.980000
939.989967	124.885000	939.989967	126.885000
949.989980	125.800000	949.989980	127.800000
959.989993	126.725000	959.989993	128.725000
969.990007	127.635000	969.990007	129.635000
979.990020	128.520000	979.990020	130.520000
989.990033	129.430000	989.990033	131.430000
999.990049	130.350000	999.990049	132.350000

Drift Curve

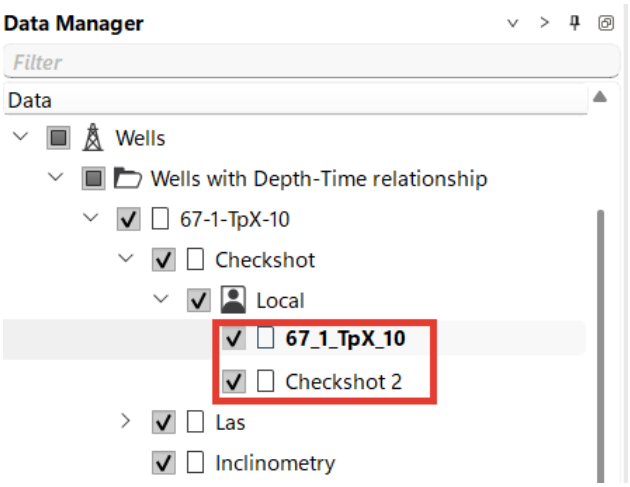
As you edit the anchors, g-Space automatically calculates a **Drift** curve, defined as the difference in two-way time (ΔTWT) between the adjusted (edited) checkshot and the original base checkshot at each depth. The drift shows how far the calibrated time-depth relationship departs from the original checkshot and is a useful quality-control indicator of the tie.

The **Drift** curve is refreshed automatically each time the checkshot is edited and is displayed in the well tie view together with the other checkshot-derived curves. It is also stored as a LAS curve (in milliseconds) under the corresponding well in the **Data Manager**, so it can be reviewed or exported like any other well log.

To delete the current version of your checkshot, select **Remove All Anchors** from the top menu. This will also update the checkshot curve.

© 2026 Geomage 2003 LTD

In g-Space, you can upload multiple checkshots. To perform a well tie, select the active checkshot in the Data Manager by double-clicking on it.

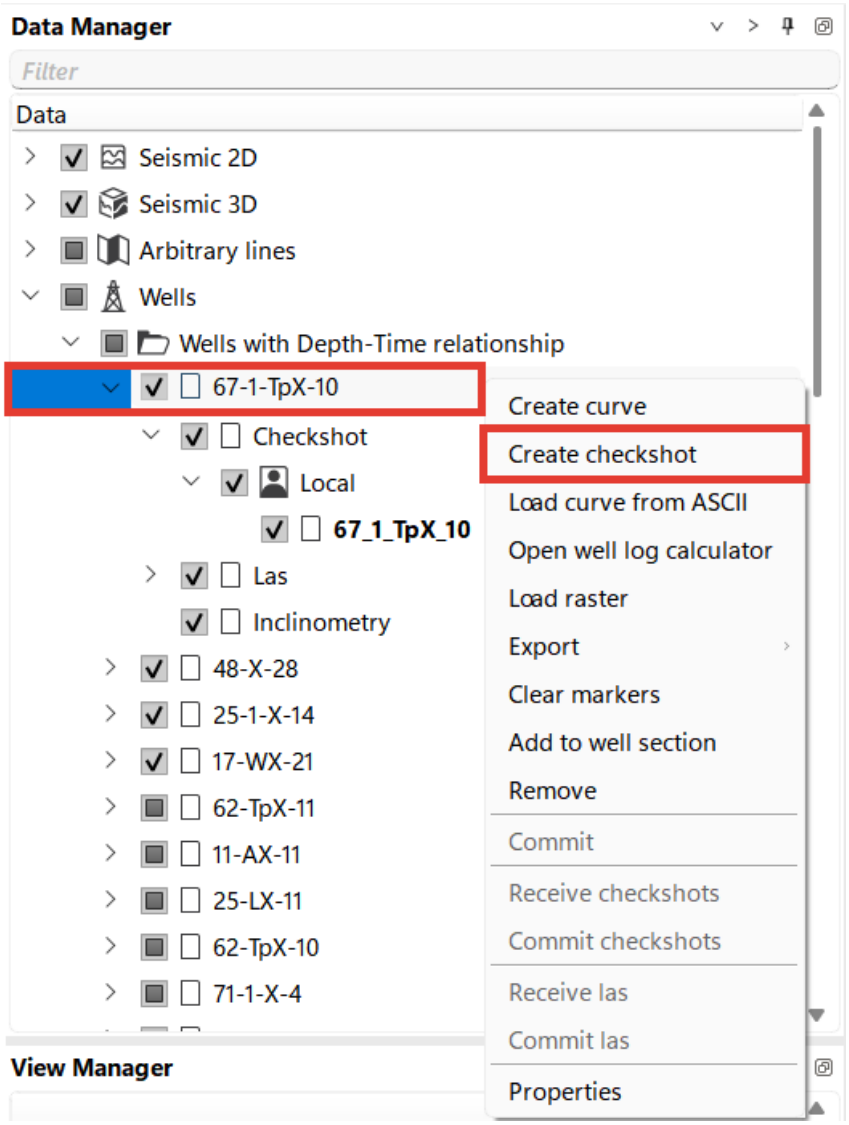


Creating a check-shot from sonic log

If the wells with Sonic logs are not loading for well-tie, please perform the following steps:

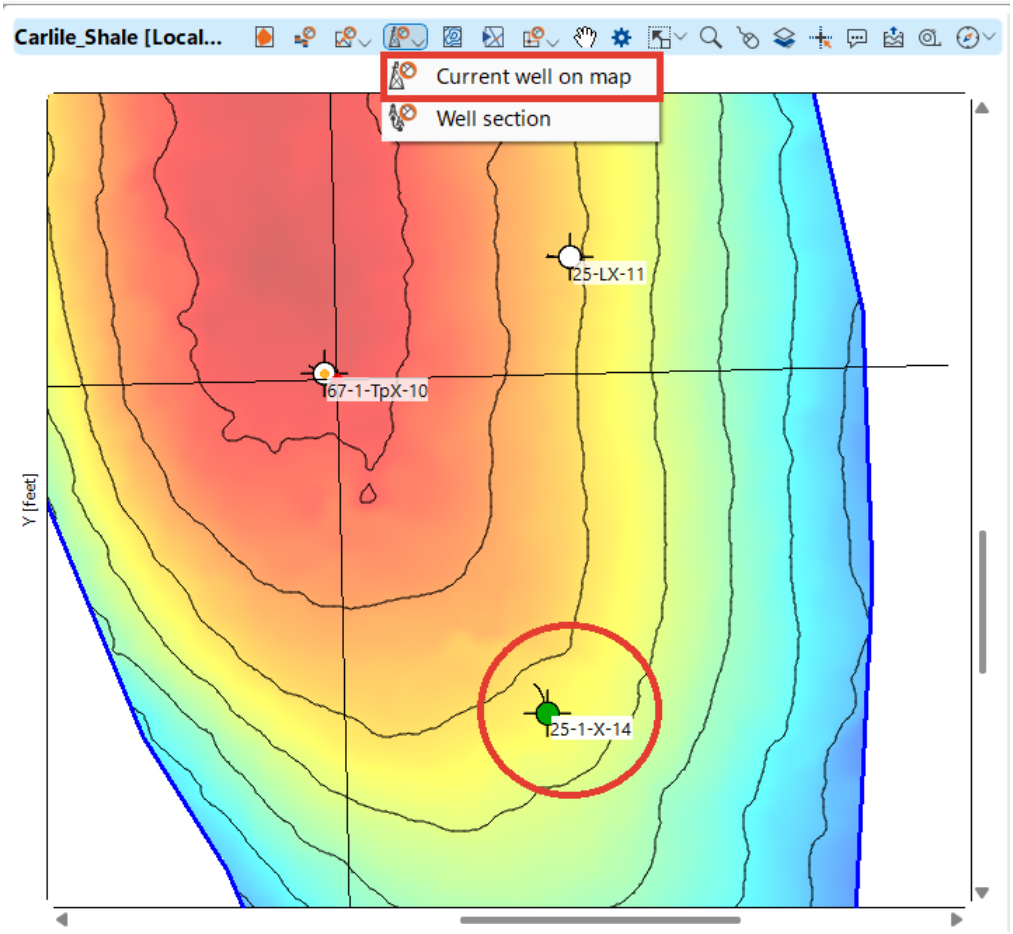
1. Create a new checkshot for this well in the Data Manager; make sure that this checkshot is active.

© 2026 Geomage 2003 LTD



2. Go to **Location map** and select "**Current well on map**" icon

© 2026 Geomage 2003 LTD



Click on the well and it will display a new window as shown below

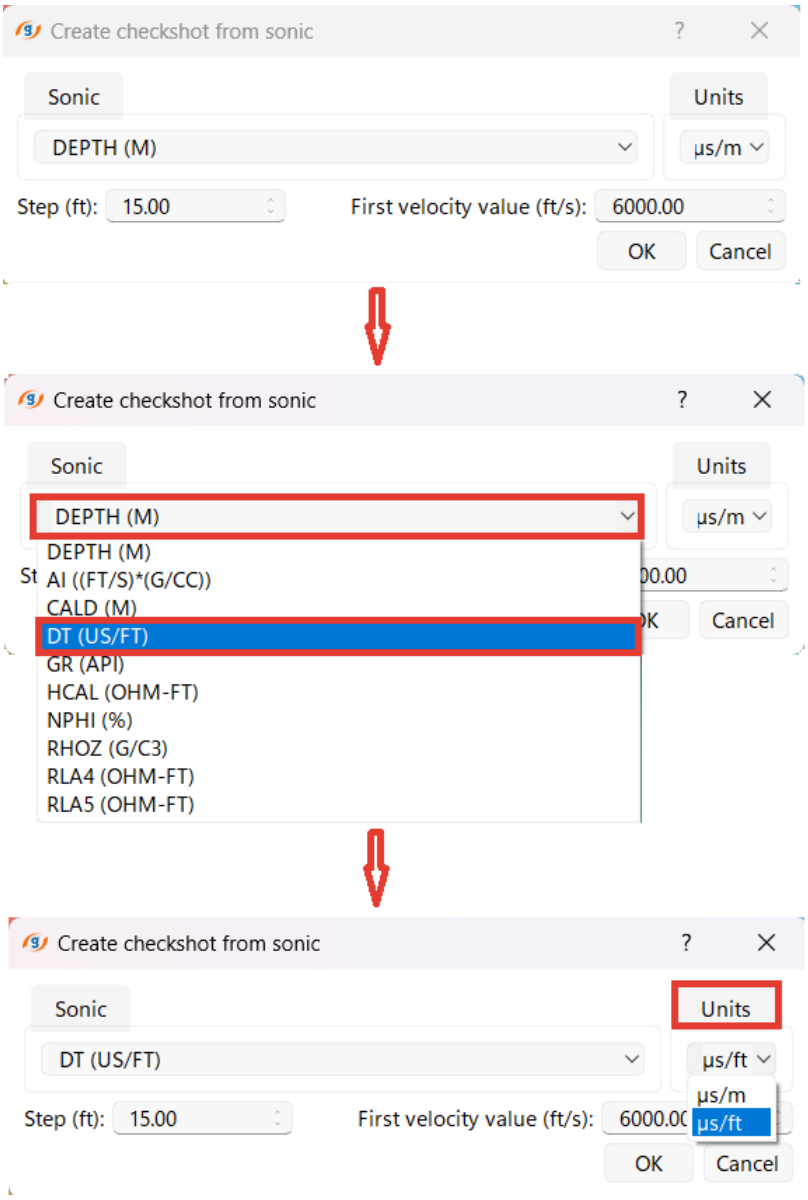
Now set the curve corresponding to the sonic from the drop-down list, in our case **DT** log, and make sure the units are correct.

Set **Step** for checkshot and **First velocity value**

Notes

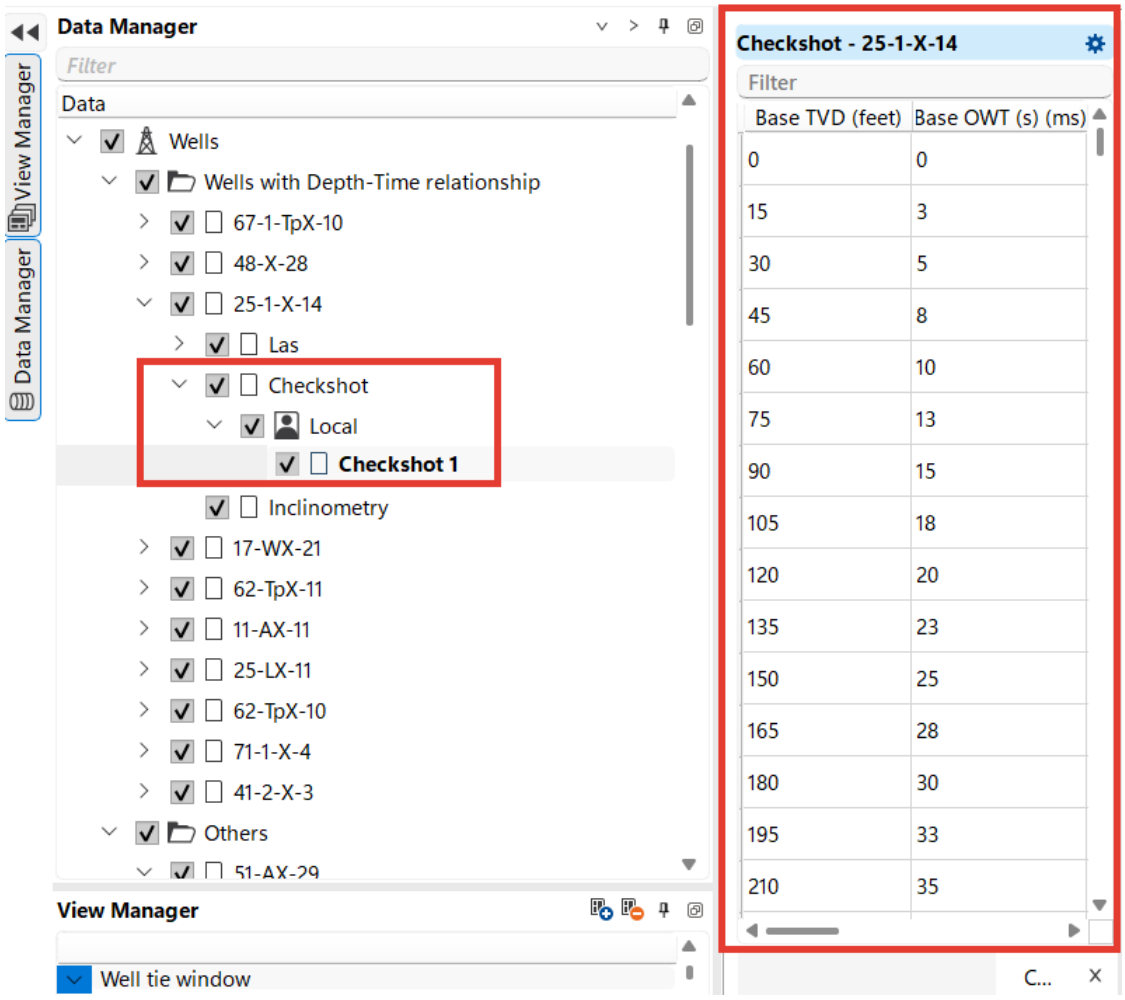
1. *The process of generating a checkshot from a sonic log involves using the sonic log data to estimate the travel time of seismic waves through the subsurface. A sonic log provides interval transit times (Δt), which is the time it takes for a sound wave to travel through a unit length of rock. By integrating these transit times over the depth of the well, we can calculate the cumulative travel time and create a synthetic checkshot.*
2. *However, the sonic log often does not cover the uppermost part of the well, leaving a gap in the data. In such cases, the user must manually set a velocity value for the first available depth in the log. This user-defined velocity serves as an approximation for the travel times through the upper layers that are not recorded in the sonic log.*

© 2026 Geomage 2003 LTD



User can see the created checkshot in the well folder in the data manager and open it as a table

© 2026 Geomage 2003 LTD



Once satisfied with the tie, finalize it by saving the synthetic seismogram and the adjusted Time-Depth curve.

Notes

1. *Datum Level Consideration. The datum level represents a reference point, usually the mean sea level or another fixed elevation, from which depths are measured. Ensuring that all wells and seismic data are tied to the same datum level is essential to avoid misalignments.*
2. *Understanding Different Data Scales and Resolutions. Be aware of potential mismatches in resolution when correlating seismic and well data. For example, subtle geological features visible in well logs may not be as apparent in seismic data.*
3. *Reviewing Log Data for Errors. Log data may contain errors, such as spikes, shifts, or inconsistent readings, which can lead to incorrect synthetic seismograms and false reflections. A thorough review of the log data by a petrophysicist is recommended before proceeding with the well-to-seismic tie.*

© 2026 Geomage 2003 LTD

4. *Calibration with Checkshot Data. Always calibrate your well logs against checkshot data, if available. Checkshots provide more accurate time-depth information and help correct any discrepancies between the log-derived and actual time-depth relationships. Cross-verify the time-depth data with multiple sources, such as VSP (Vertical Seismic Profiling) data, to ensure consistency and accuracy.*

9.5 Seismic picking

The **Seismic Picking** tools in g-Space let interpreters digitise features on 2D and 3D seismic data — horizons, faults, mute zones and individual control points. Picks are tied to the active seismic view and immediately become project objects in the Data manager 124 so they can drive downstream workflows such as velocity modelling, volumetric calculations and well-tie QC.

This guide describes the picking tools using the 3D seismic volume and 2D profiles from the Demo project. *Subscribe to our channel and watch the video on how to trace horizons in g-Space on* YouTube.

What can be picked

- **Horizons** — continuous reflector surfaces traced across seismic lines. Output is stored as a `GHorizon` object and used by map creation, velocity modelling and depth-conversion tools.
- **Faults** — fault sticks interpreted on individual lines; collected into a `GFault` object.
- **Mutes** — amplitude-masking regions used by gather-conditioning and AVO workflows.
- **Checkpoint picks** — individual control points used for well-tie QC and precision picking, visualised as Vista items in the current or all layers.

Starting a picking session

1. Open a Seismic view 240 (2D) or 3D view 244 showing the data to interpret.
2. Select the feature type from the picking toolbar: horizon, fault, mute, or checkpoint.
3. Pick interactively with the mouse — single click to add a point, drag to trace a continuous segment, double-click to close the feature.
4. Quality-control indicators (picking coverage, number of phase jumps for horizons) are updated live in the view's status area.

© 2026 Geomage 2003 LTD

Quality control

Horizon picks are evaluated on two QC metrics computed automatically:

- **Picking coverage** — percentage of traces along the line where the horizon was actually picked.
- **Phase jumps** — count of 180° discontinuities, used to spot pick-to-pick mis-ties.

Tips

- Start interpretation on the most continuous line in the survey, then propagate to adjacent lines — g-Space snaps the pick to the nearest consistent reflector.
- Use the precision picking mode to force the pick onto the exact sample under the cursor when auto-snap is not desired.
- Review picks in the 3D view before building maps — cross-line consistency is easier to spot there than on individual 2D sections.

See Also

- Seismic view 240
- 3D view 244
- AI Faults Wizard 642
- Map creating 724

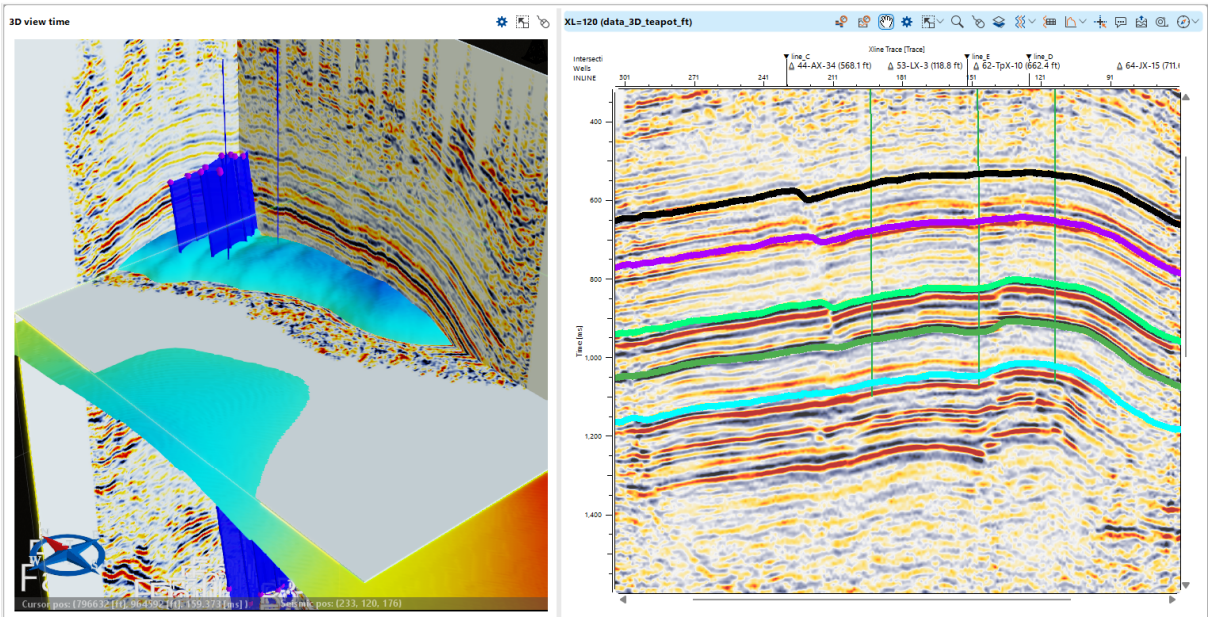
9.5.1 Seismic data visualization

Effective visualization helps in identifying and tracing geological features, which are essential for building accurate geological models.

In g-Space, several types of data visualization can be utilized, each serving a specific purpose in the interpretation process. These include 2D seismic 589 sections for detailed layer-by-layer inspection, 3D volumes 599 for a more comprehensive spatial overview, and attribute maps 105 that reveal specific characteristics such as amplitude or coherence anomalies.

The availability of multiple visualization modes in g-Space, such as interactive zooming, rotation, and filtering, enhances the user's ability to accurately interpret seismic data. Users can switch between these views through the View Manager 189, enabling seamless navigation and analysis across different data types. This flexibility supports both large-scale structural analysis and finer-scale reservoir characterization.

© 2026 Geomage 2003 LTD



When tracing horizons, use the flattening feature in g-Space to align your seismic data along a selected horizon. This makes it easier to identify continuous features and avoid misinterpretation.

Remember that seismic data can sometimes contain noise or artifacts. Use filters and attribute analysis to enhance the clarity of your data before making final interpretations.

Notes

Without effective visualization, it is challenging to interpret complex seismic datasets, as the relationships between geological features can easily be overlooked.

Visualization also plays a role in attribute analysis. Seismic attributes, such as amplitude, frequency, or coherence, provide additional insights into rock properties and fluid content. Displaying these attributes in visual formats, such as attribute maps or color-coded 3D volumes, allows interpreters to analyze subtle changes in the seismic response, leading to a more detailed and accurate geological interpretation.

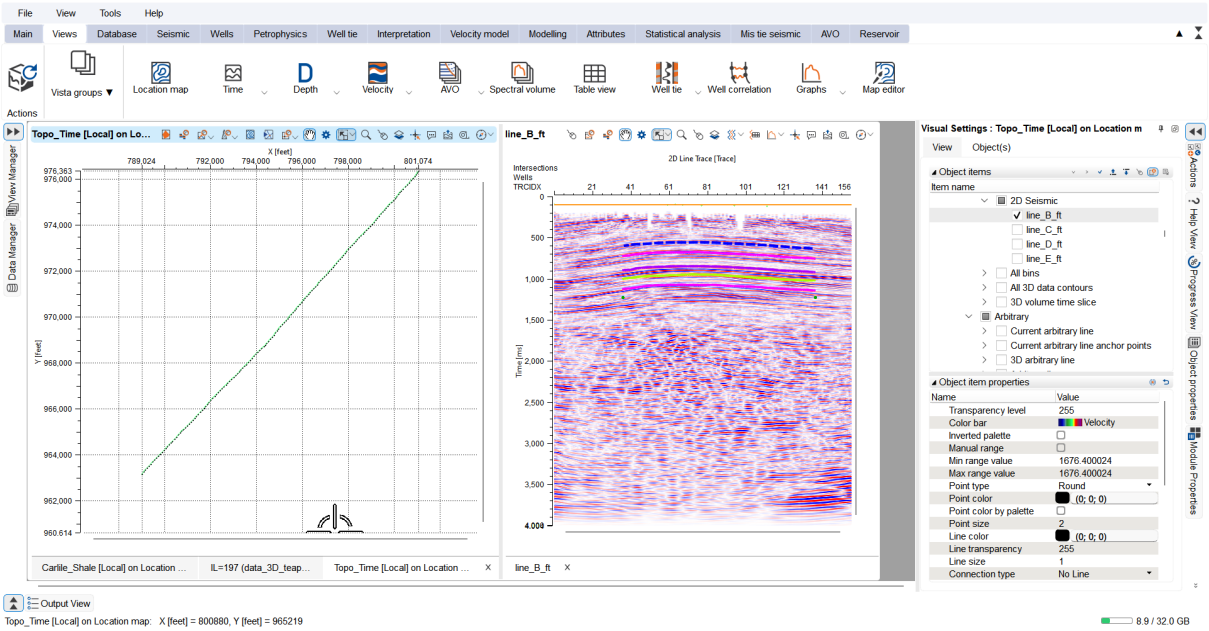
In g-Space, these theoretical principles are put into practice through tools that support dynamic data manipulation, real-time updates of seismic displays, and interactive exploration of multi-dimensional datasets.

© 2026 Geomage 2003 LTD

9.5.1.1 2D data

2D seismic data can be visualized in g-Space both on a section or within a 3D window, depending on the interpretation needs. Let's explore how seismic data is displayed in section and review visual settings that aid in picking geological horizons, faults, and other subsurface structures.

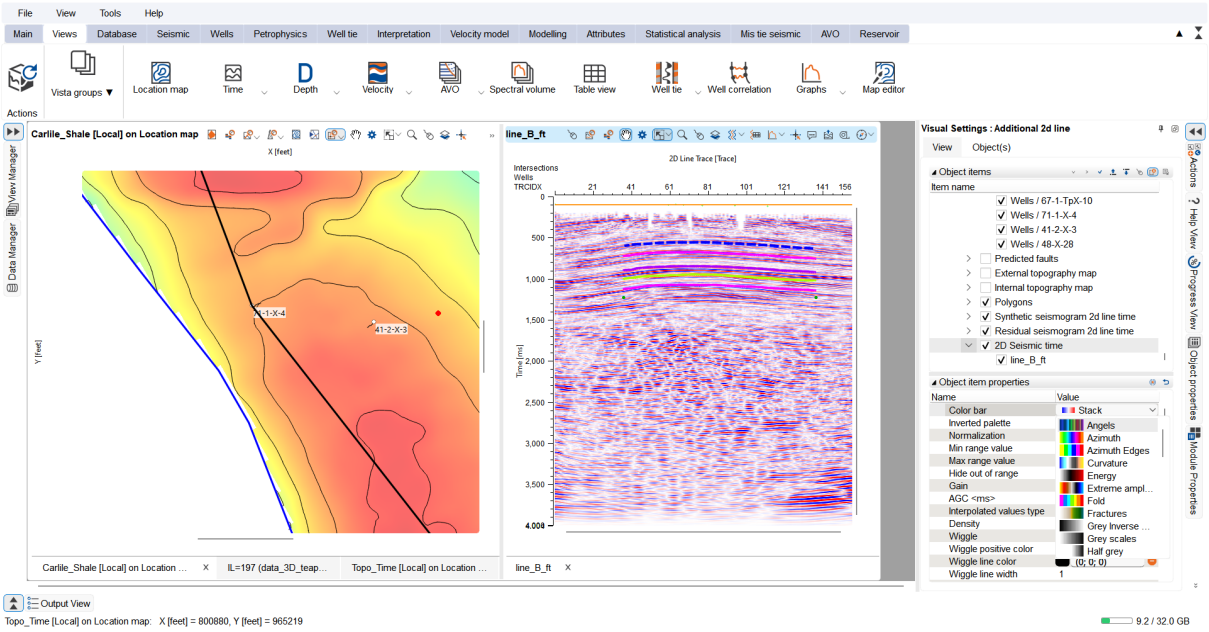
In this example, a workspace is prepared in View Manager 189 to visualize 2D seismic profiles. The workspace consists of a *Location Map* window on the left and a *2D Time Section* window on the right. These windows are synchronized for seamless interpretation. The active profile is highlighted in green, and you can easily switch between different seismic profiles, which are visualized both on the map and as seismic sections.



Users can control the appearance of the seismic lines and points by adjusting their properties in the Visual Settings 206 panel. Below is an example of a visual setup of a seismic section.
Click on ⚙ and the Panel Visual Settings will be opened.

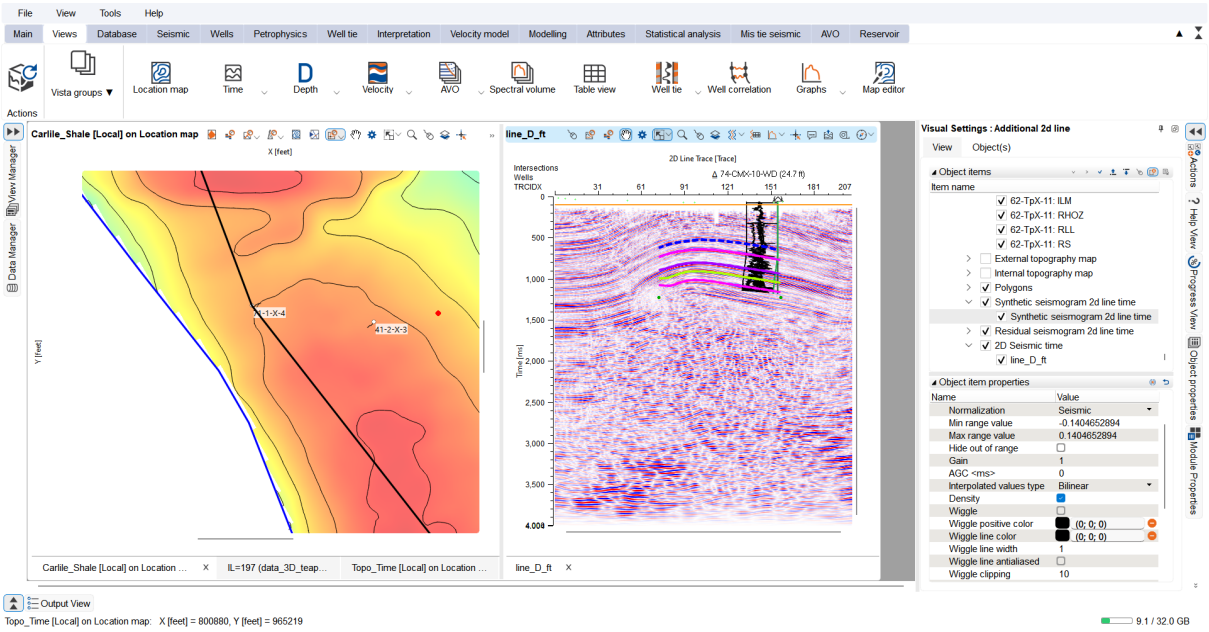
There is a list of data that can be visualized in the View. Here, we have shown 3D horizons and fault interpretations on 2D data. In **Object items** for a **Location map**, select **2D Seismic**. In a time section, select **2D Seismic time**; in a depth section, select **2D Seismic depth**. Converted data is listed as **Converted 2D Seismic time** or **Converted 2D Seismic depth**, respectively. Select the seismic item to set its color palette and other display options in **Object item properties**.

© 2026 Geomage 2003 LTD



The example of well data visualization on a seismic section in the image below shows the well trajectory, log, synthetic trace, and markers. You can adjust the fill and line colors, the thickness of the synthetic trace, choose to display it as wiggles or in color, and modify the amplitude for the most effective visualization.

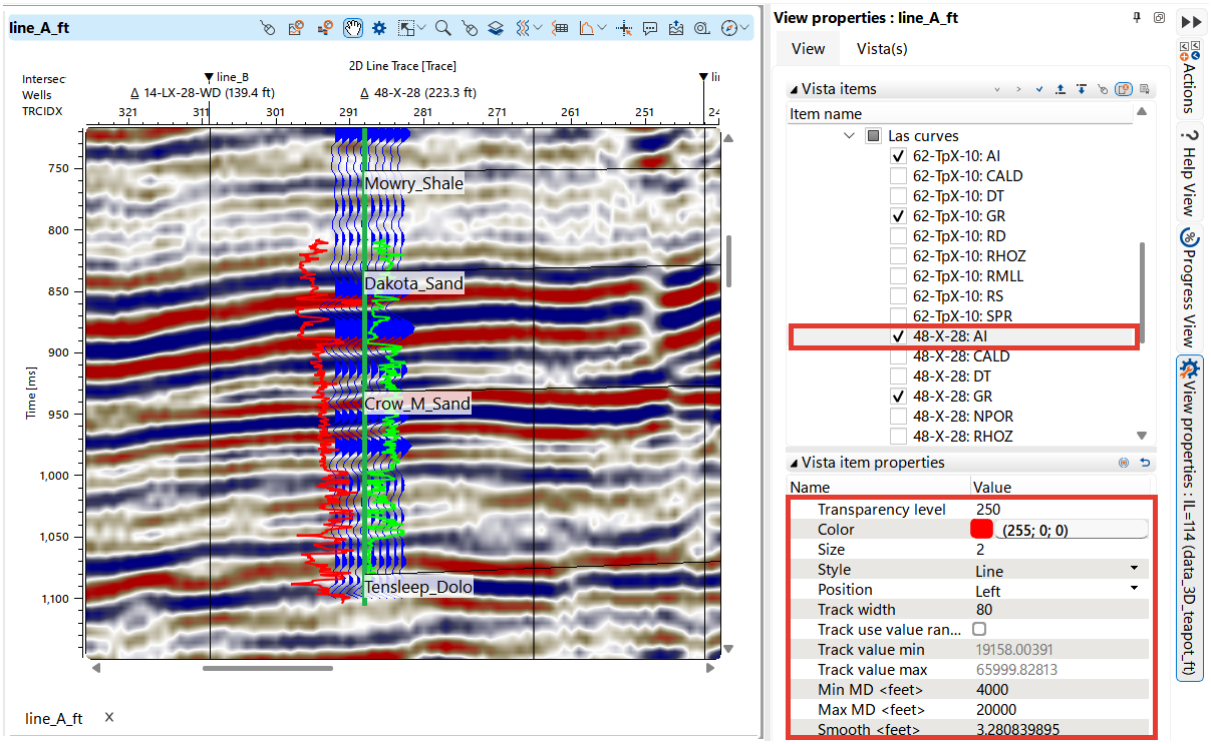
The part of the 2D line displayed in the seismic section is highlighted on the Location Map.



© 2026 Geomage 2003 LTD

The user has the option to display all logging curves or select specific ones. Settings can be adjusted for all curves by clicking on **Las Curves**, or for individual curves by clicking directly on the curve name.

In this image, the Acoustic Impedance and Gamma Ray curves have been customized with selected color and thickness, are displayed to the left and right of the well trajectory, and have their tracks width and depth interval modified. Additionally, transparency settings and minimum and maximum value adjustments are available.



Using the **Arbitrary Line** Tool in g-Space

Arbitrary Line is particularly useful in the following scenarios:

- **Comparing 2D and 3D Data:** This tool is essential for comparing and assessing the quality of different seismic surveys. By creating custom lines that traverse both 2D profiles and the 3D seismic volume, users can directly compare the resolution and continuity of seismic reflections, leading to a better understanding of the subsurface.
- **Correlating Wells:** The arbitrary line is helpful for correlating wells, especially when you need to understand the structural and lithofacies features of the interwell space. By drawing a line that connects multiple wells, you can visualize how stratigraphic horizons and structural features change between wells. A detailed description is

© 2026 Geomage 2003 LTD

available in the 3D Data 599 chapter, as these tasks are most commonly handled with 3D data.

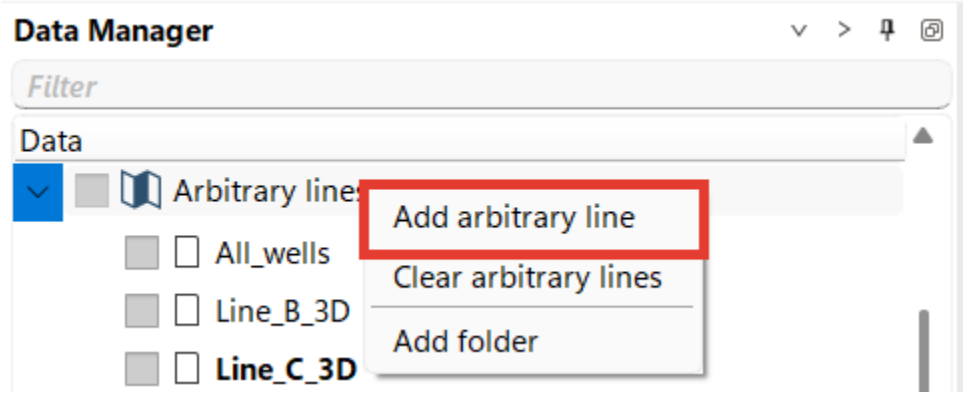
- **Tying Directional Wells:** When dealing with directional or deviated wells, the arbitrary line allows you to create cross-sections that follow the wellbore trajectory. This aids in accurately tying the well data to seismic data, ensuring a more precise integration of well and seismic interpretations.
- **Analyzing Geological Structure:** For fields with complex geological features, such as the anticline fold in the Demo project, the arbitrary line tool is particularly useful for analyzing geological structures both along and across the fold axis. Given that the fold has a curved structure relative to the geometry of the seismic survey, arbitrary lines enable a more flexible and accurate examination of the fold's features, which might not align with standard seismic lines.

Notes

Use the Arbitrary Line tool to investigate complex geological structures that might not be apparent on standard inline or crossline sections.

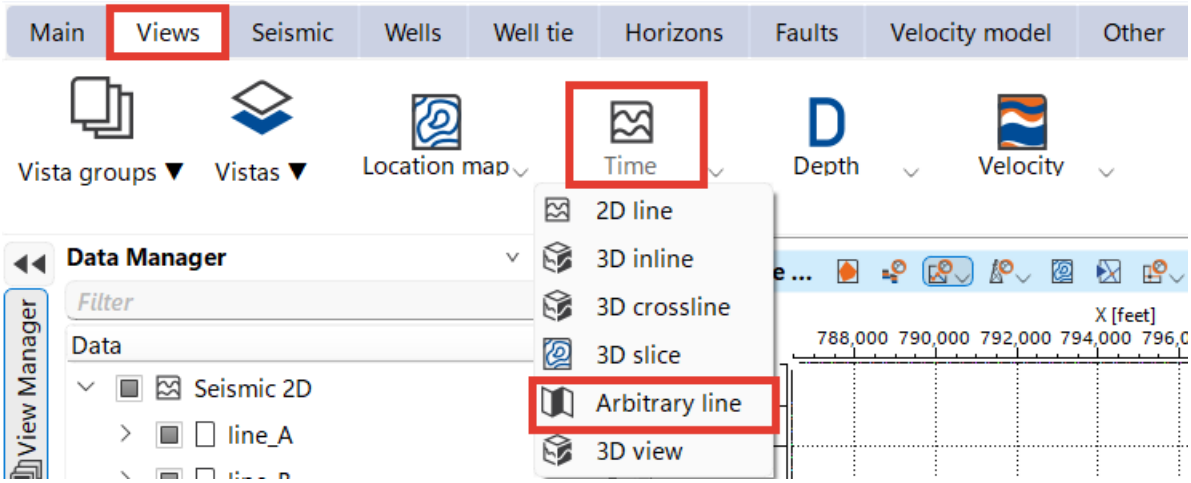
To create an **Arbitrary line** from 2D profile segments, follow these instructions:

In Data manager click on Arbitrary lines folder and choose **Add arbitrary line**, a pop-up will open, name the line

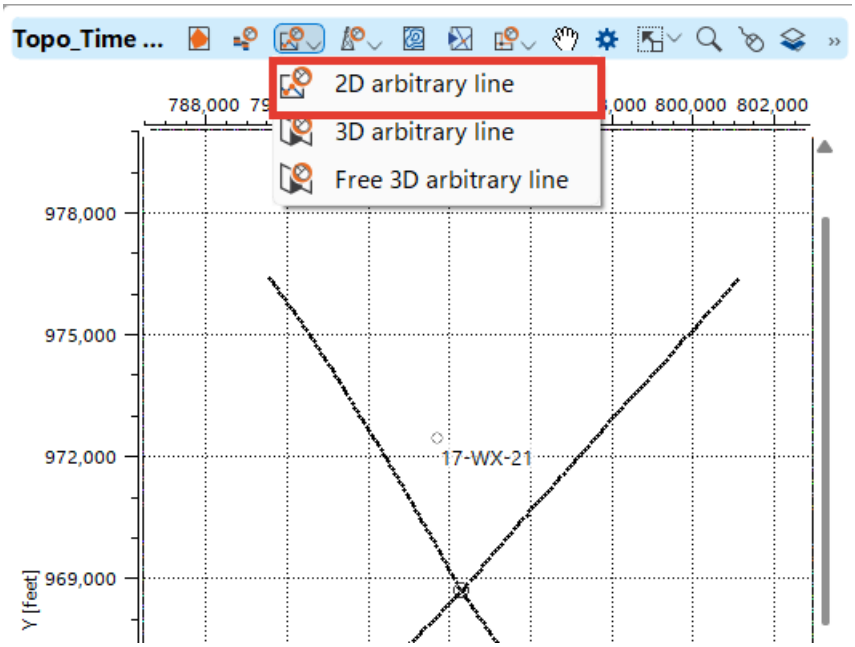


In the **View bar** select **Time**, then **Arbitrary line**. Now there are two Views in Workspace - **Location Map** and empty **Arbitrary line view**.

© 2026 Geomage 2003 LTD



On Map View click Arbitrtary line and choose 2D arbitrary line.

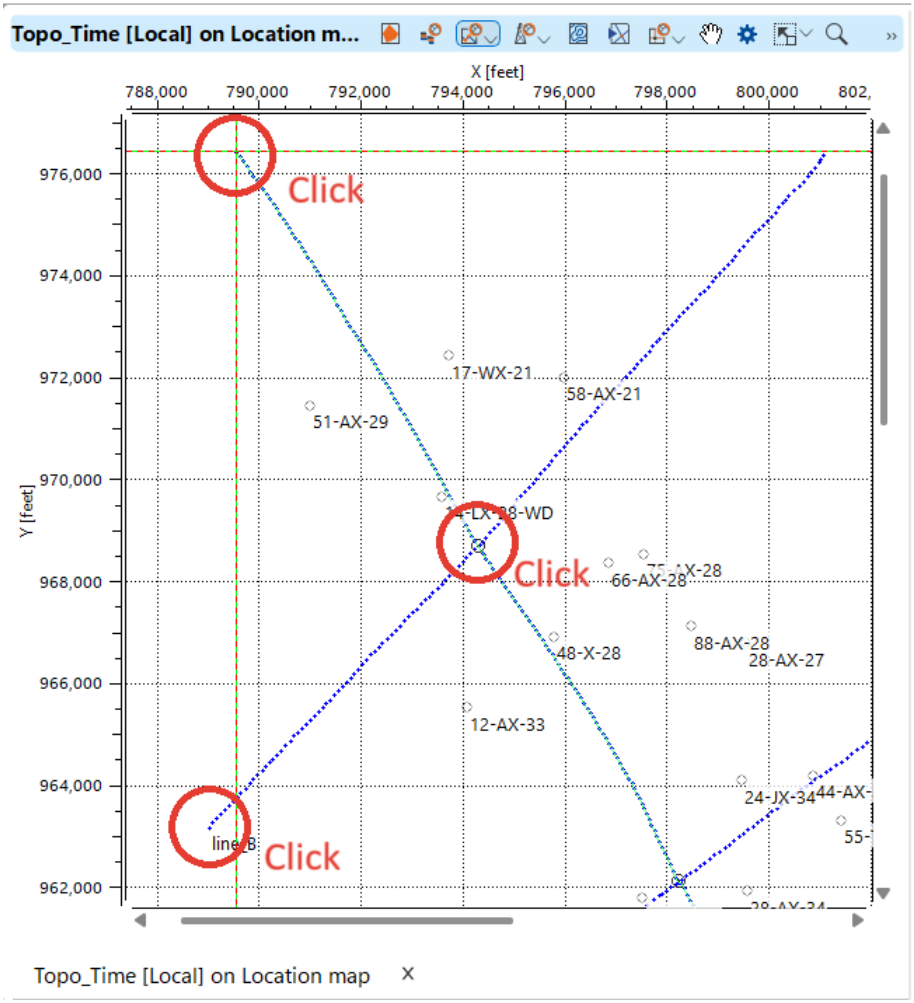


The cursor will change to an active cross. By clicking on the desired 2D line segments, the user will create the geometry of the arbitrary line.

To complete the curve creation, double-click the left mouse button at the last point.

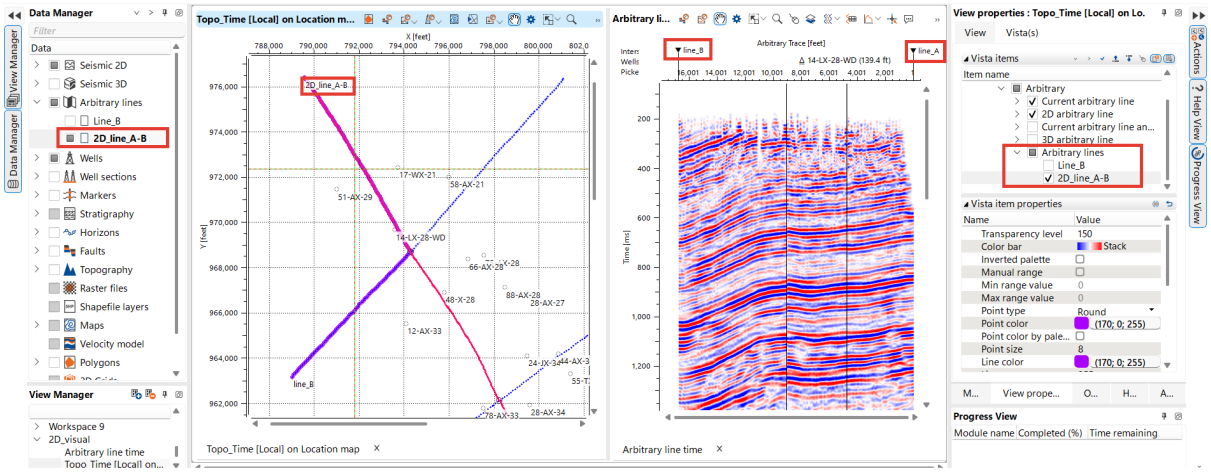
To remove a point from a created arbitrary line, select it by holding down the right mouse button.

© 2026 Geomage 2003 LTD

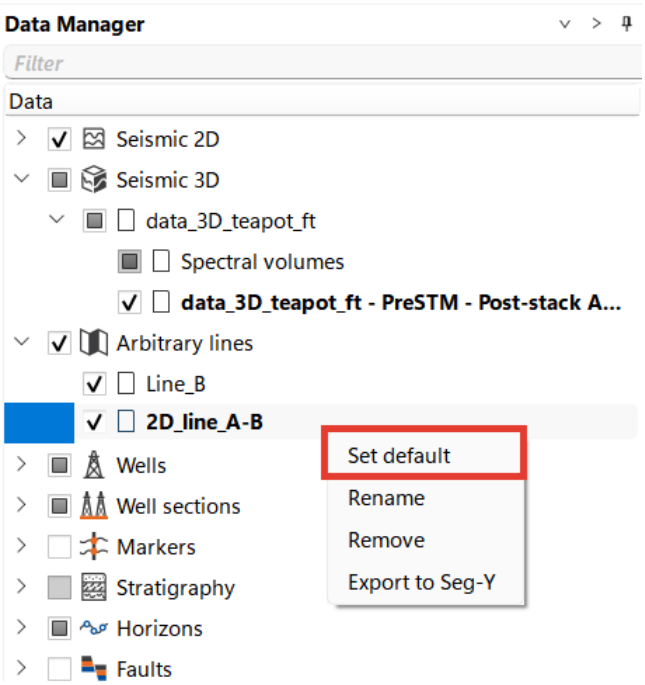


A new arbitrary line will be created, you will see it both on the **Map** and in the **Arbitrary line** window, the visualization settings can be set in the Visual Settings, in the case below the name of the line is added as well as a color is set.

© 2026 Geomage 2003 LTD



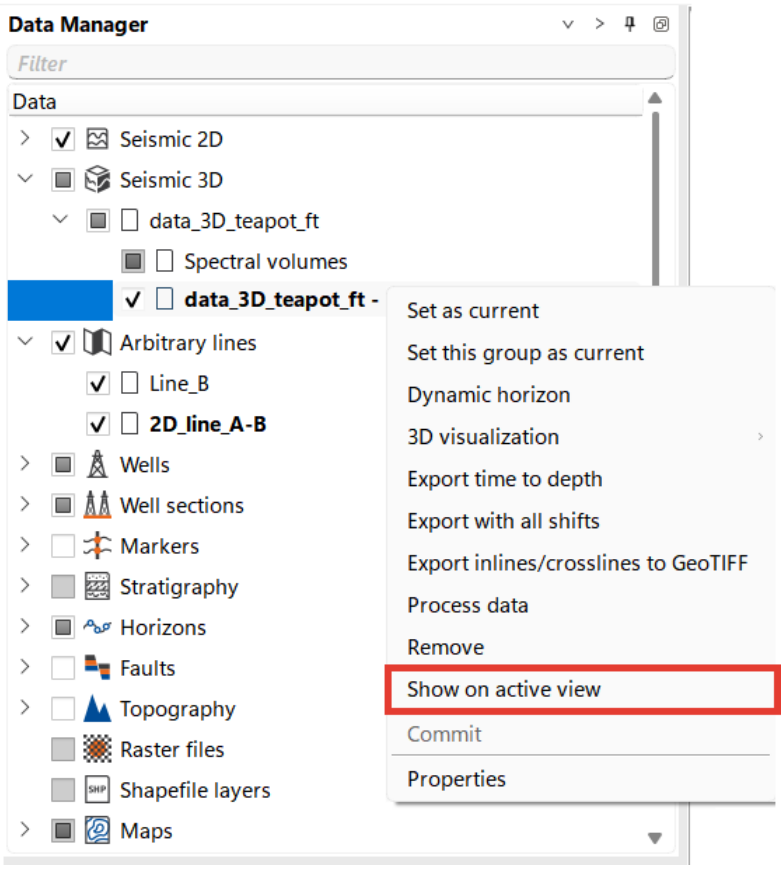
For comparing 2D and 3D Data, make the 2D Arbitrary line (or profile) active in the Data Manager.



Create 2 Arbitrary lines, go to **Views** - **Time/Depth** scales, then choose **Arbitrary line**.

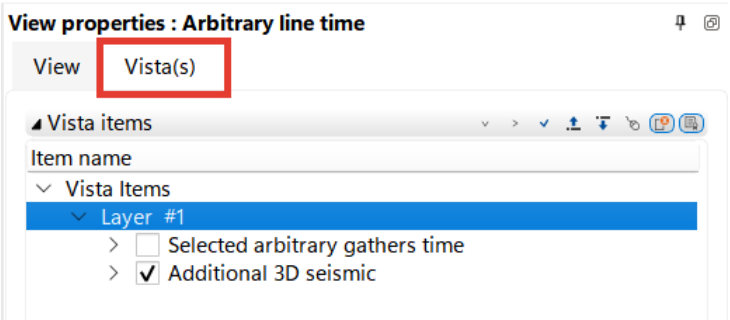
Next, right-click on the 3D dataset you want to visualize and select **Show on active view** from the drop-down menu.

© 2026 Geomage 2003 LTD



Alternatively, use the checkbox in the **Data Manager**: select the checkbox next to a 3D dataset that the active view does not display yet, and the dataset is added to the view as **Additional 3D seismic**. Clear the checkbox to remove the dataset from the view. The checkbox works this way in **2D line**, **3D inline**, **3D crossline** and **Arbitrary line** views in time and depth.

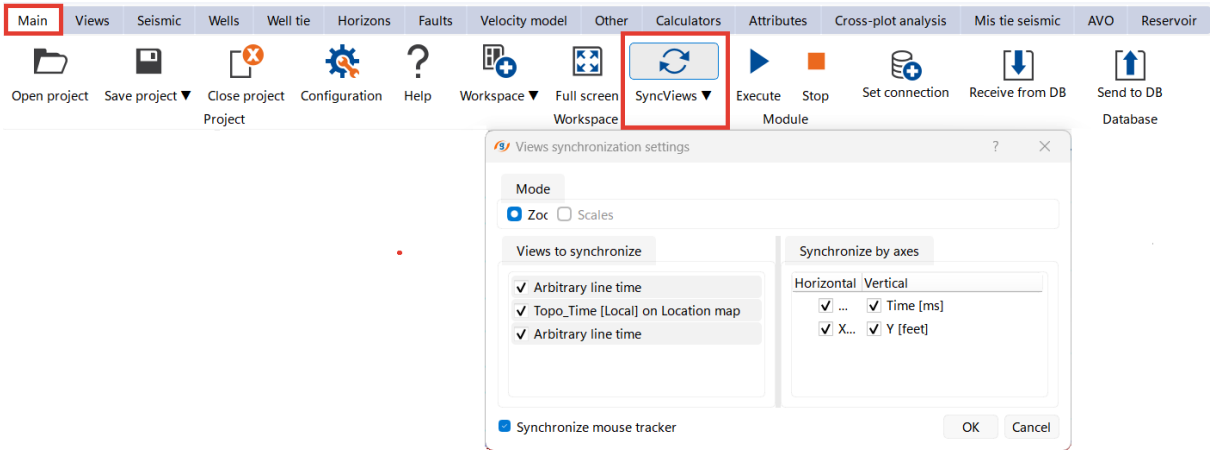
Open the Visual Settings for the Arbitrary line. In the **Object(s)** tab, select Arbitrary line (2D data at the example below) for one Arbitrary line View and **Additional 3D seismic** for the other view, respectively.



A full description of the settings can be found in Seismic View 240

© 2026 Geomage 2003 LTD

To synchronize views, go to the Main bar and click SyncViews. Additional settings are available in the drop-down menu.



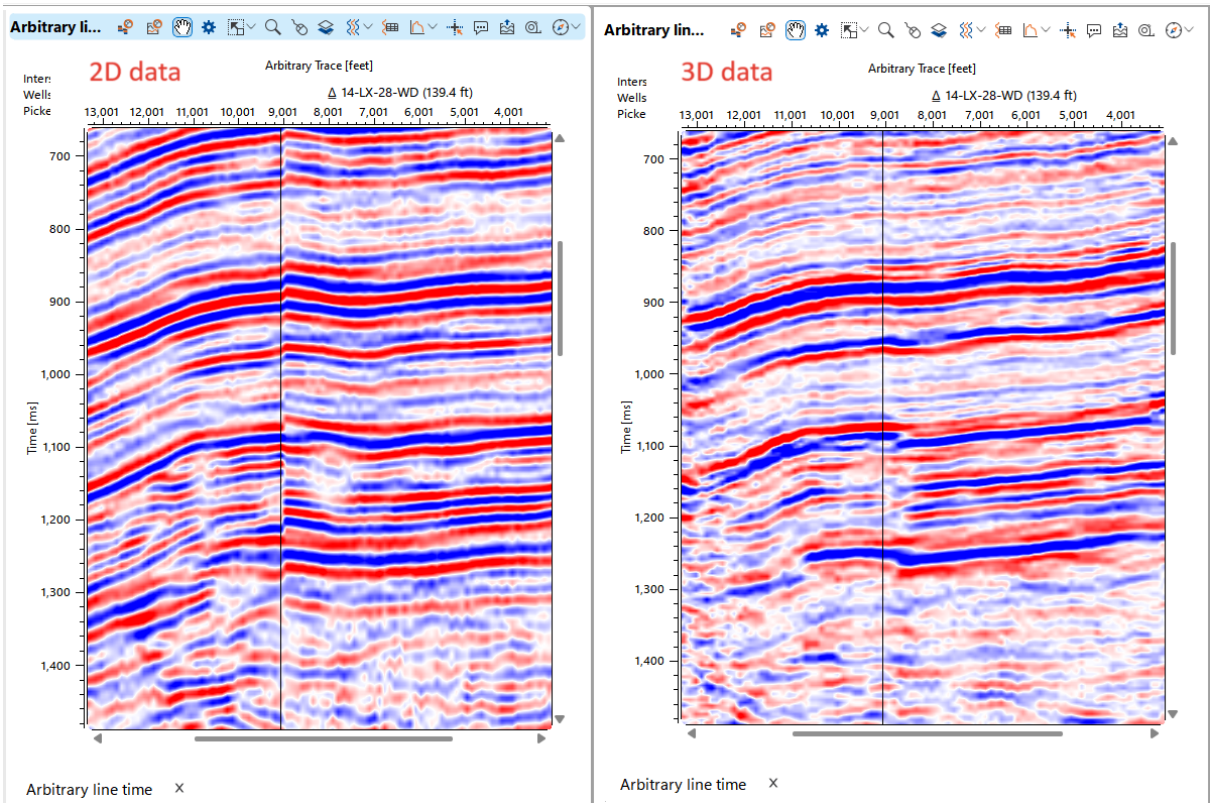
You can now visualize and compare the generated arbitrary lines. In our example, the image below illustrates the comparison results.

This visual comparison helps highlight how arbitrary lines drawn from 2D and 3D data can show differences in the continuity and resolution of seismic events. It's important to analyze both representations to better understand the subsurface geology.

*The **2D data** appears smoother with well-defined reflectors, especially in the shallower section. The reflectors are more continuous.*

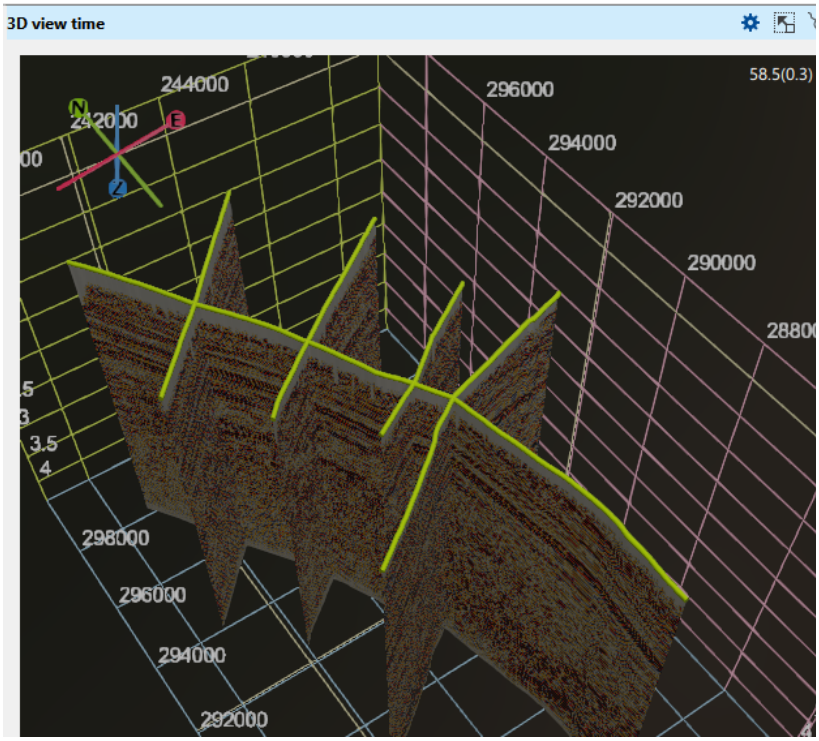
*The **3D data** in the lower section shows greater variability and more detailed structure, which is common for 3D seismic data, as it provides additional subsurface information.*

© 2026 Geomage 2003 LTD



To display 2D profiles in 3D, select 3D view 244 in time or depth scale depending on your data.

© 2026 Geomage 2003 LTD



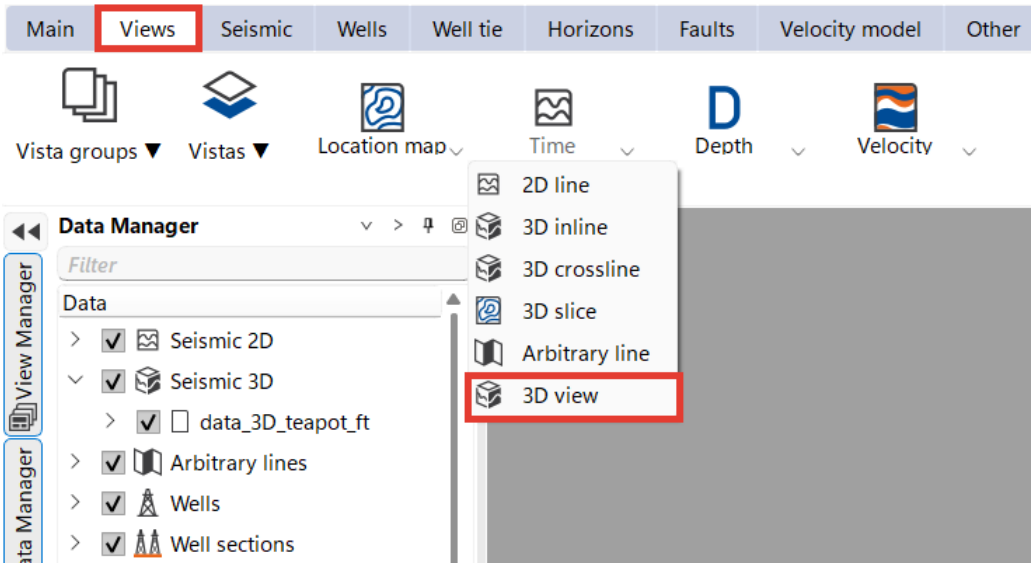
A full description of the settings can be found in 3D View 244

9.5.1.2 3D data

3D seismic data can be visualized in g-Space both on a section or within a 3D window, depending on the interpretation needs. Let's explore how seismic data is displayed in section and review visual settings that aid in picking geological horizons, faults, and other subsurface structures.

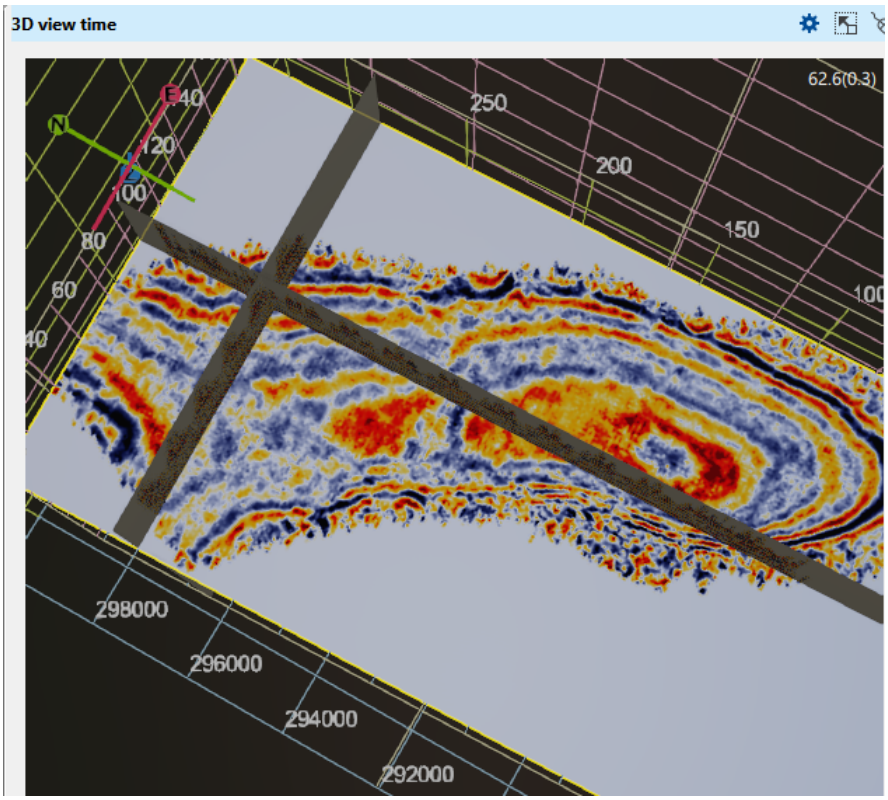
To create a 3D view, go to the Views 233 in the Ribbon Bar 33, select a Time or Depth scale and a 3D view.

© 2026 Geomage 2003 LTD



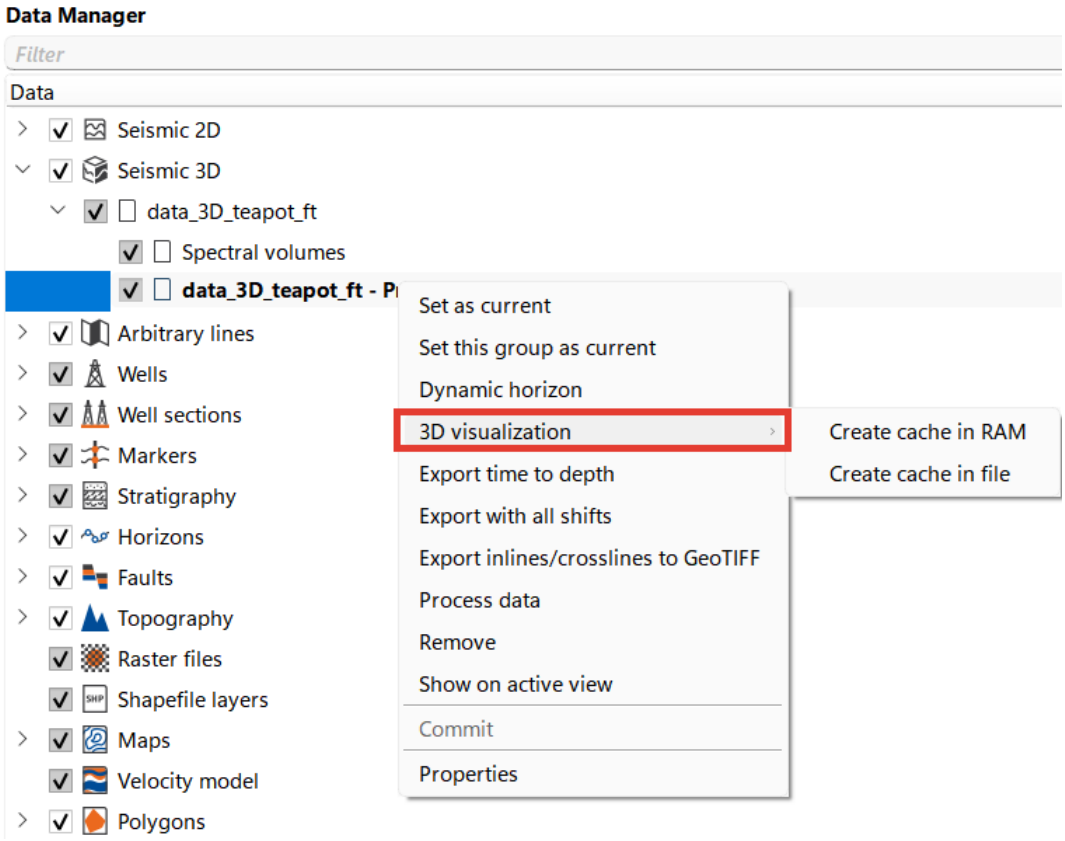
By default, a new Time or Depth 3D view hides ordinary project items. In the Data Manager, select the checkboxes for the items you want to display. An already enabled cached 3D volume remains visible in the new view.

You can view an entire 3D volume or seismic sections and time/depth slices in 3D View.



© 2026 Geomage 2003 LTD

Right click on the 3D volume you want to display, select **3D visualization** and create a cache for it in a RAM or file. You can find detailed explanation which path to choose in 3D View 244



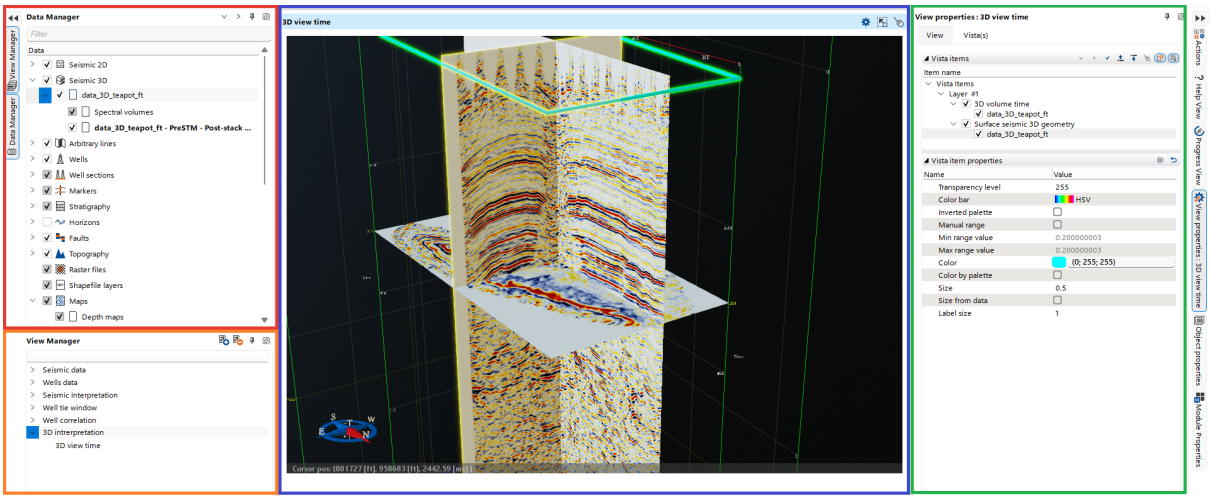
Within the **View Manager** (highlighted in orange) users have the capability to construct a customized workspace by combining multiple views.

The highlighted blue **Workspace** displays active views. To add a desired view, navigate to the Views 233 in the Ribbon bar. To close a View, click on the 'x' at the bottom of the view next to its name and confirm your choice.

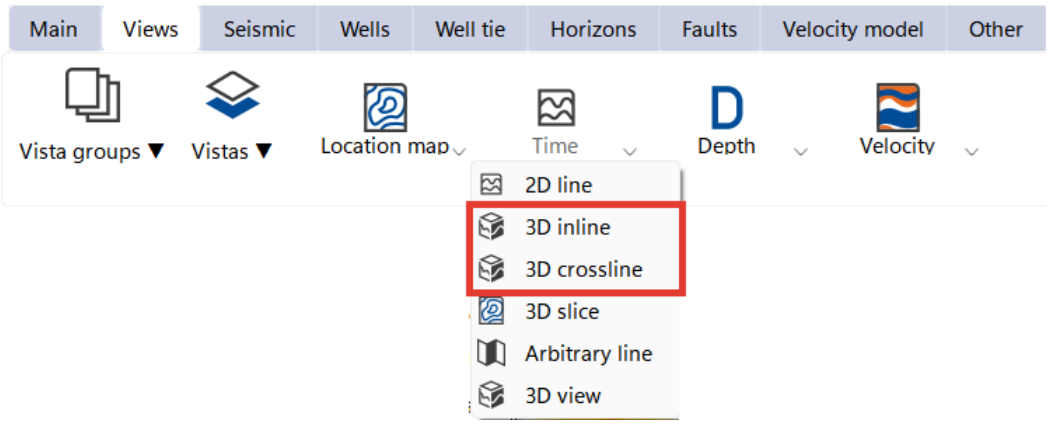
On the right-hand side, you can find the Settings area. For each active element in the Data Manager, its visualization settings will be displayed in the Visual Settings 206. To switch to another object, click on it in the **Data Manager**.

Detailed settings and Hot keys described in chapter 3D View 244

© 2026 Geomage 2003 LTD

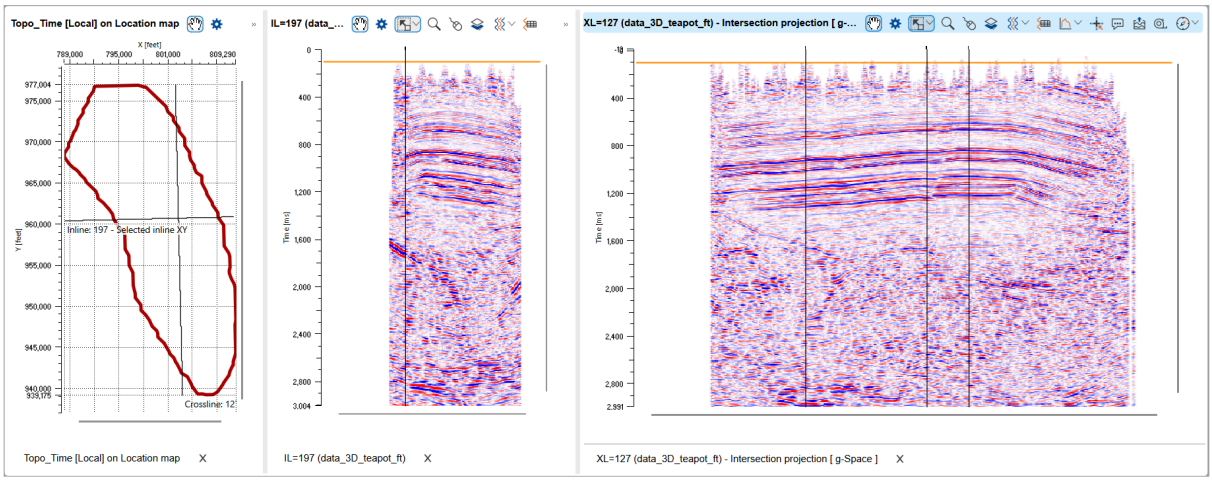


The primary visualization parameters when working with 3D seismic cubes are inlines and crosslines. To create new inlines or crosslines, navigate to the time or depth domain under the Views 233 and select **3D inline** or **3D crossline**



The image below shows the inline and crossline sections, as well as the 3D volume on the Map view with the indicated sections and the intersection point (highlighted in red on map and in blue lines on the sections). It is convenient to visualize inline and crossline views and synchronize them using the Synch feature in the Main Bar 35.

© 2026 Geomage 2003 LTD



Open the Module Properties 197 panel on the right. Here, you can set the step for switching between inlines and crosslines, as well as view or specify the current value that will be displayed in the view.

Hot keys are available:

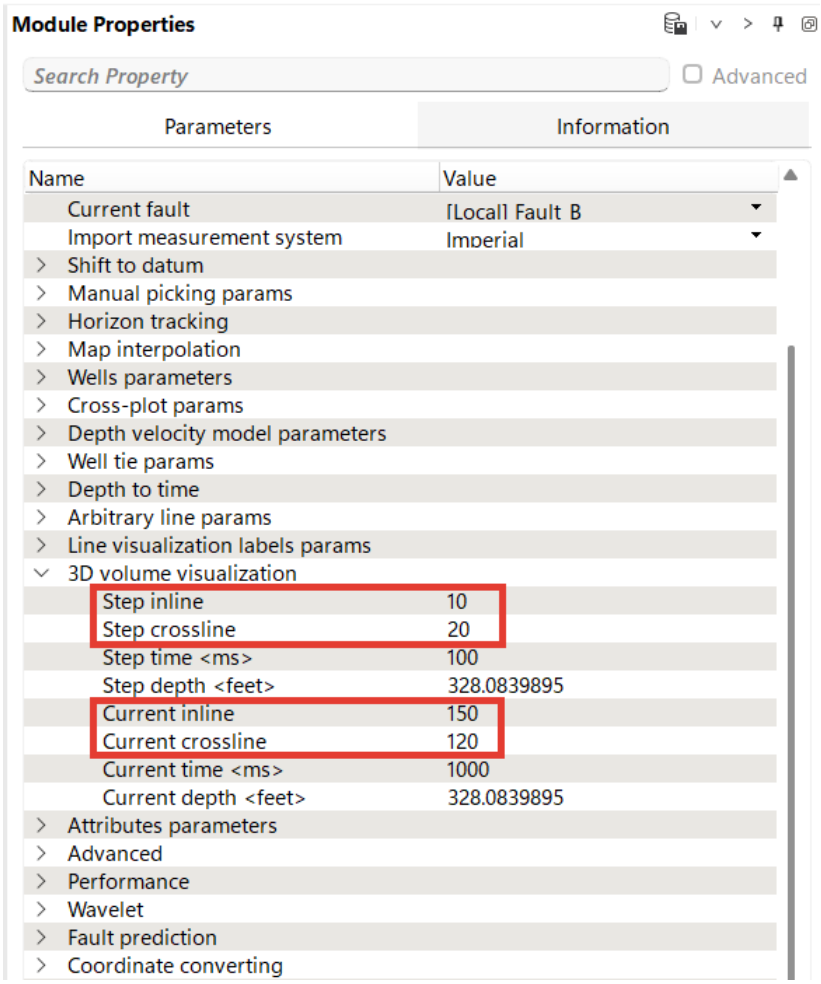
Up and **Down** move the active section up and down in the set scale

Ctrl+Left and **Ctrl+Right** move crosslines with a set step

Ctrl+Up and **Ctrl+Down** move inlines with a set step

Or **Page Up** and **Page Down** buttons move both inlines and crosslines in the active view

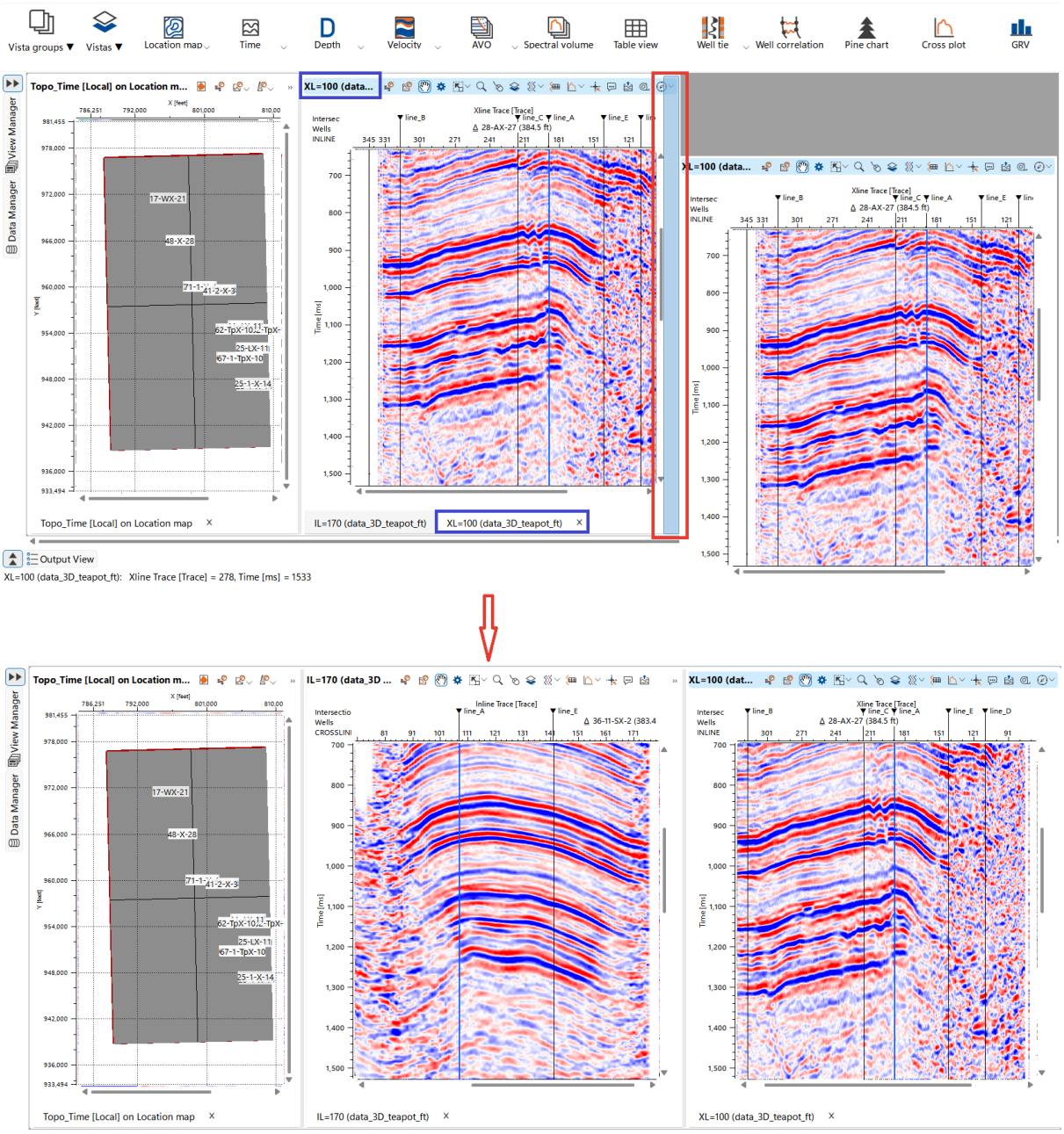
© 2026 Geomage 2003 LTD



To arrange multiple windows vertically in the workspace, click and drag the view tab (highlighted in blue) to either the top or bottom edge of the window. As you drag, a blue vertical field (highlighted in red) will appear, indicating where the window will be placed. Release the mouse button to align the windows in a vertical order within the workspace.

Similarly, to arrange windows horizontally, drag the view tab to the top or bottom edge of the window. A blue highlighted box will appear, indicating the placement. When held in place, additional layout options will be available. If the entire window is highlighted in blue, you can add the current view as a new tab to the existing view.

© 2026 Geomage 2003 LTD

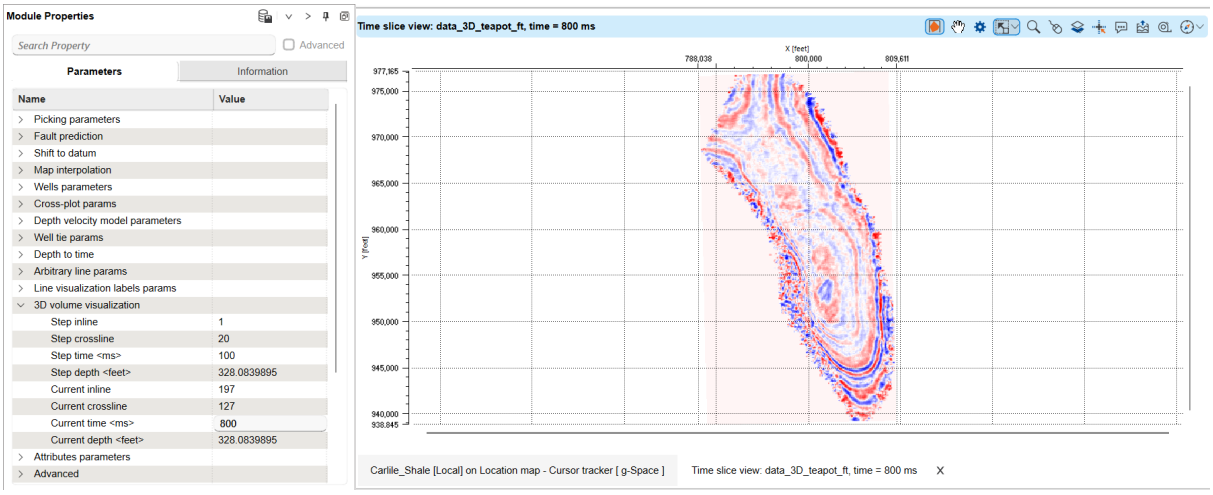


Time or depth slices offer horizontal cuts through the 3D seismic volume, displaying a snapshot of the subsurface at a specific time or depth level. This visualization is particularly useful for analyzing lateral continuity of geological features like channels or reefs.

To generate a time or depth slice in g-Space, go to the Views Bar 38 and select either Time 3D Slice or Depth 3D Slice. Define the specific time or depth level in Module Properties, and adjust the visualization settings to enhance clarity in Visual Settings 206.

© 2026 Geomage 2003 LTD

The slice view is named after the slice it displays: its title and its tab read **Time slice view: <seismic volume name>, time = <value> <unit>** in the time domain and **Depth slice view: <seismic volume name>, depth = <value> <unit>** in the depth domain, where the value is the current slice level rounded to a whole number and written in the project's time or depth unit. The name is rewritten whenever the level changes, so the level of the displayed slice can be read from the view itself.

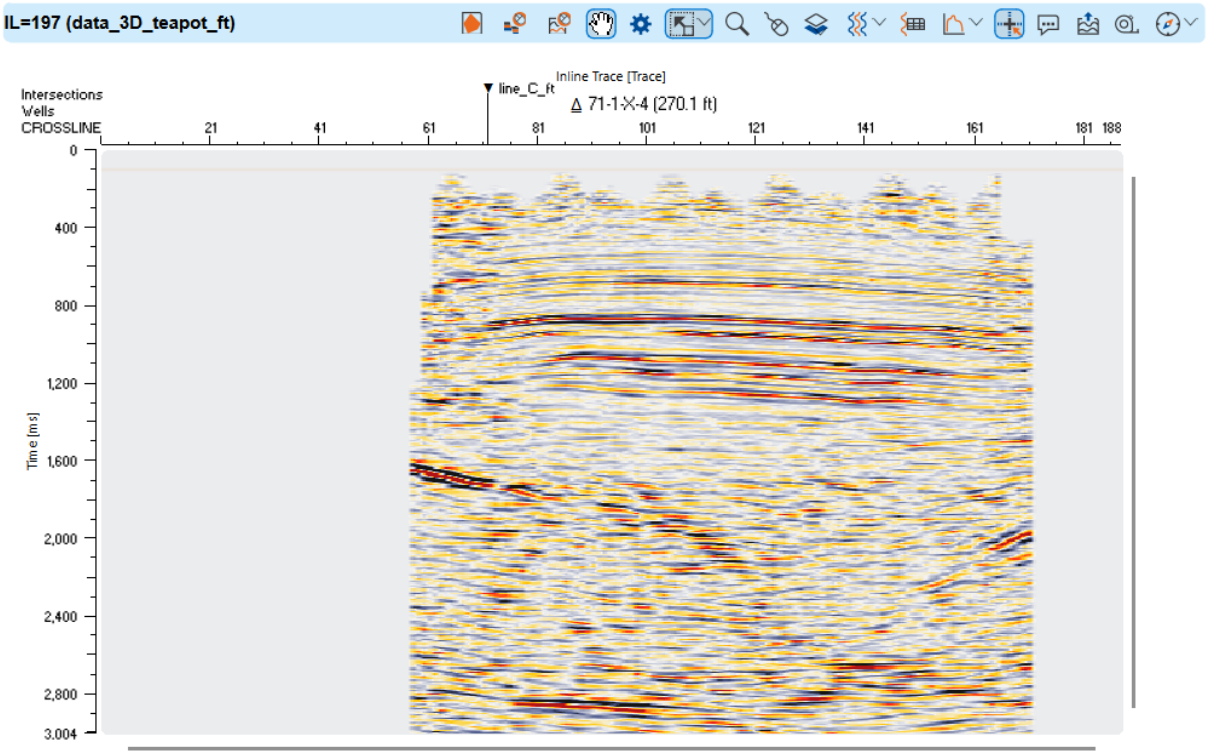


Use hotkeys **Ctrl** + **"+"** and **Ctrl** +**"-"** to switch up and down the seismic volume

Using the **Up, Down, Left** and **Right** keys you can move the image within the view.

Every time and depth section view - **3D inline**, **3D crossline**, **arbitrary line** and **2D line** - also draws the level of the current slice as a solid orange horizontal line. It is called **Current time slice** in the time domain and **Current depth slice** in the depth domain.

© 2026 Geomage 2003 LTD



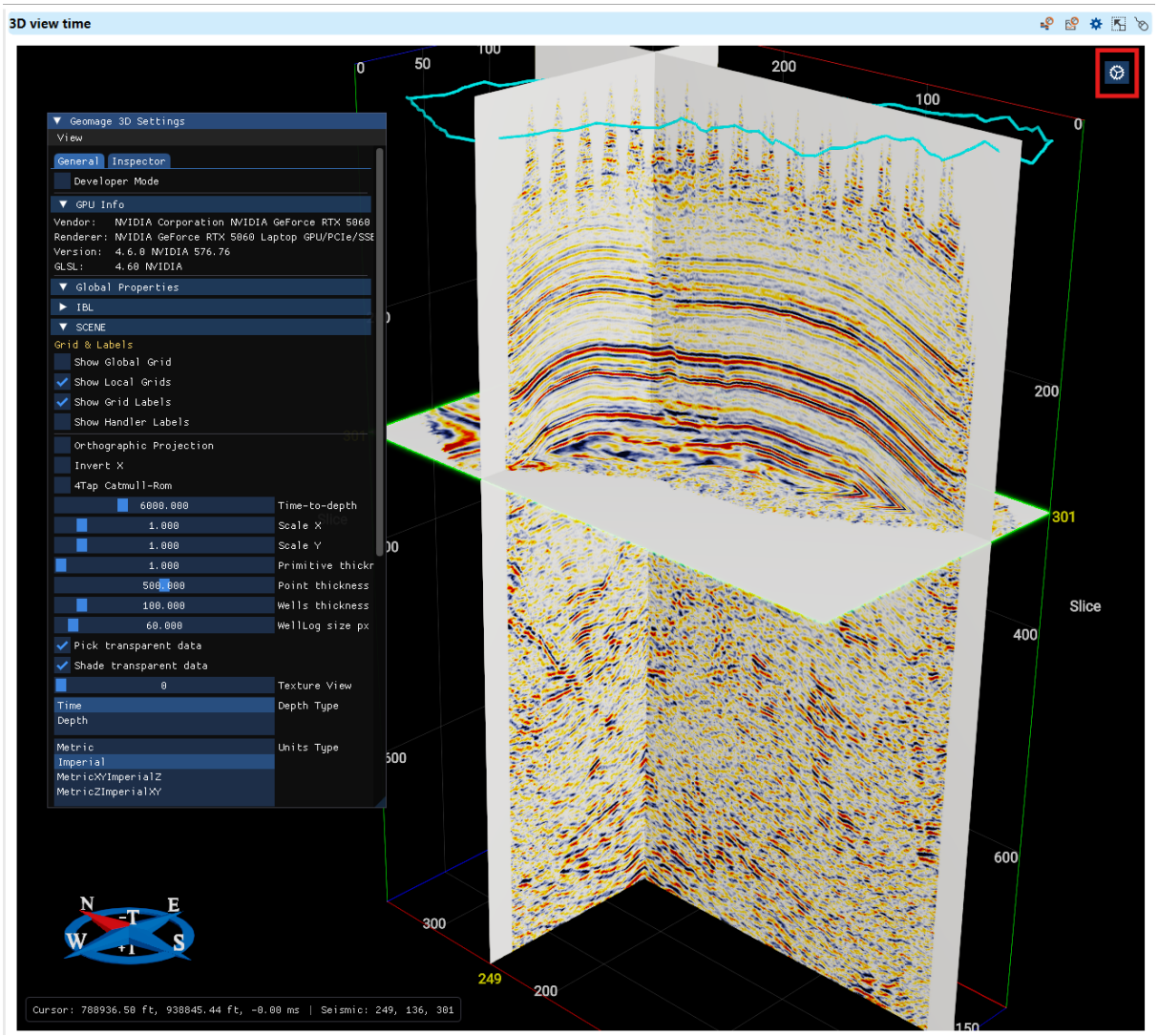
The line is always active, so you do not have to switch on a control item first. When you point at it, the cursor changes to a vertical resize cursor and the line is emphasized; drag it up or down to change the current time or depth slice. The slice view, the current slice value in Module Properties 197, the lines on the other sections and the slice plane in 3D View 244 follow the drag continuously.

The line also follows the level when you change it elsewhere - by entering a value in Module Properties 197, by using the **Ctrl** + **"+"** and **Ctrl** + **"-"** hotkeys, or by dragging the slice plane in 3D View 244.

In 3D View 244 it is convenient to visualize the seismic volume as 3 planes - inline, crossline and time slice.

The settings menu is accessible via the gear icon in the upper right corner.

© 2026 Geomage 2003 LTD



To create additional sections, use the **L** key.

While the cursor is positioned on the active section (Inline or Crossline), press **L**. A new section will be created perpendicular to the currently active one.

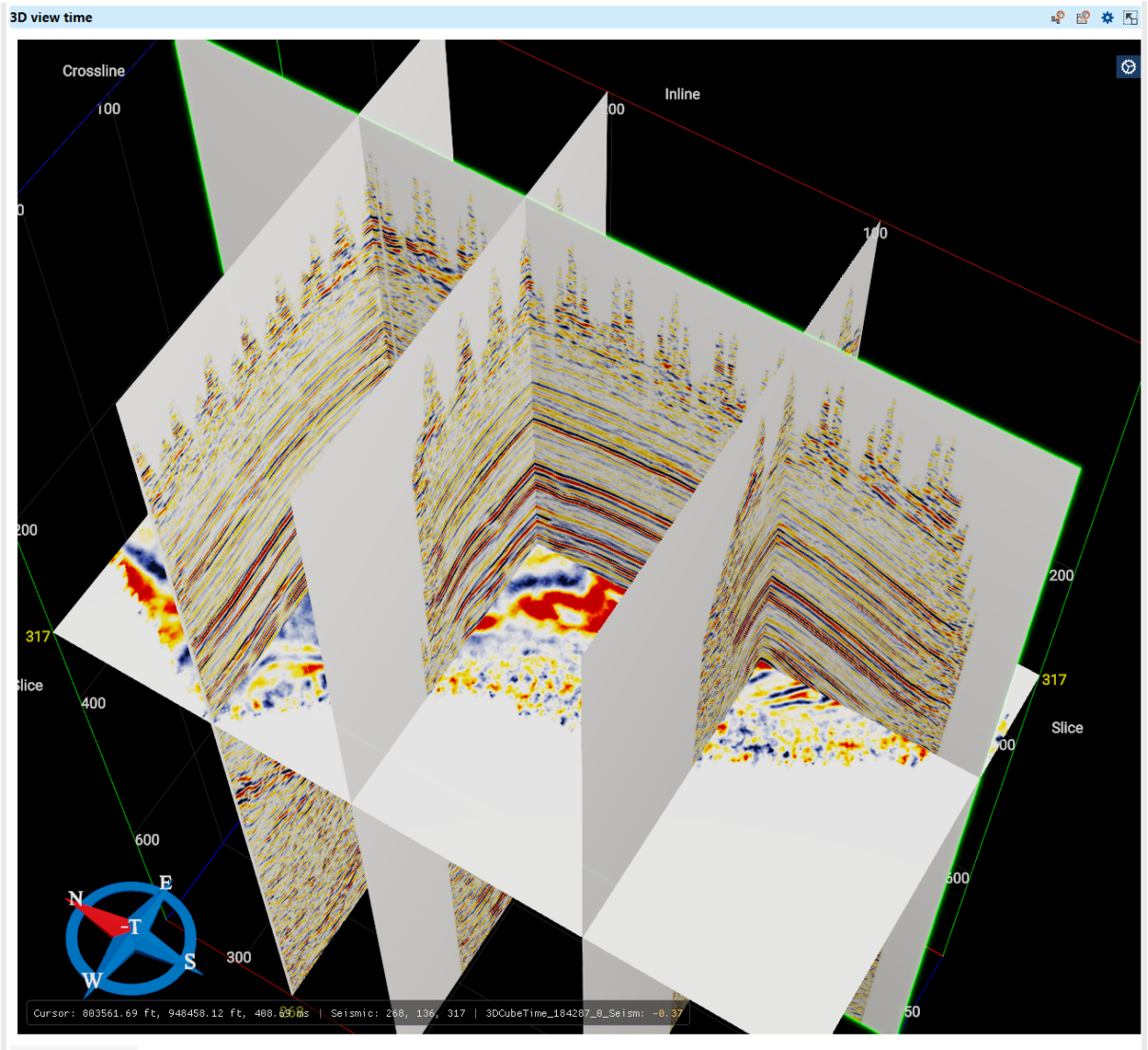
For example:

- If an **Inline** section is active, pressing **L** creates a new **Crossline** at the cursor position.
- If a **Crossline** section is active, pressing **L** creates a new **Inline** at the cursor position.

The new section is generated at the exact location where you click within the active view.

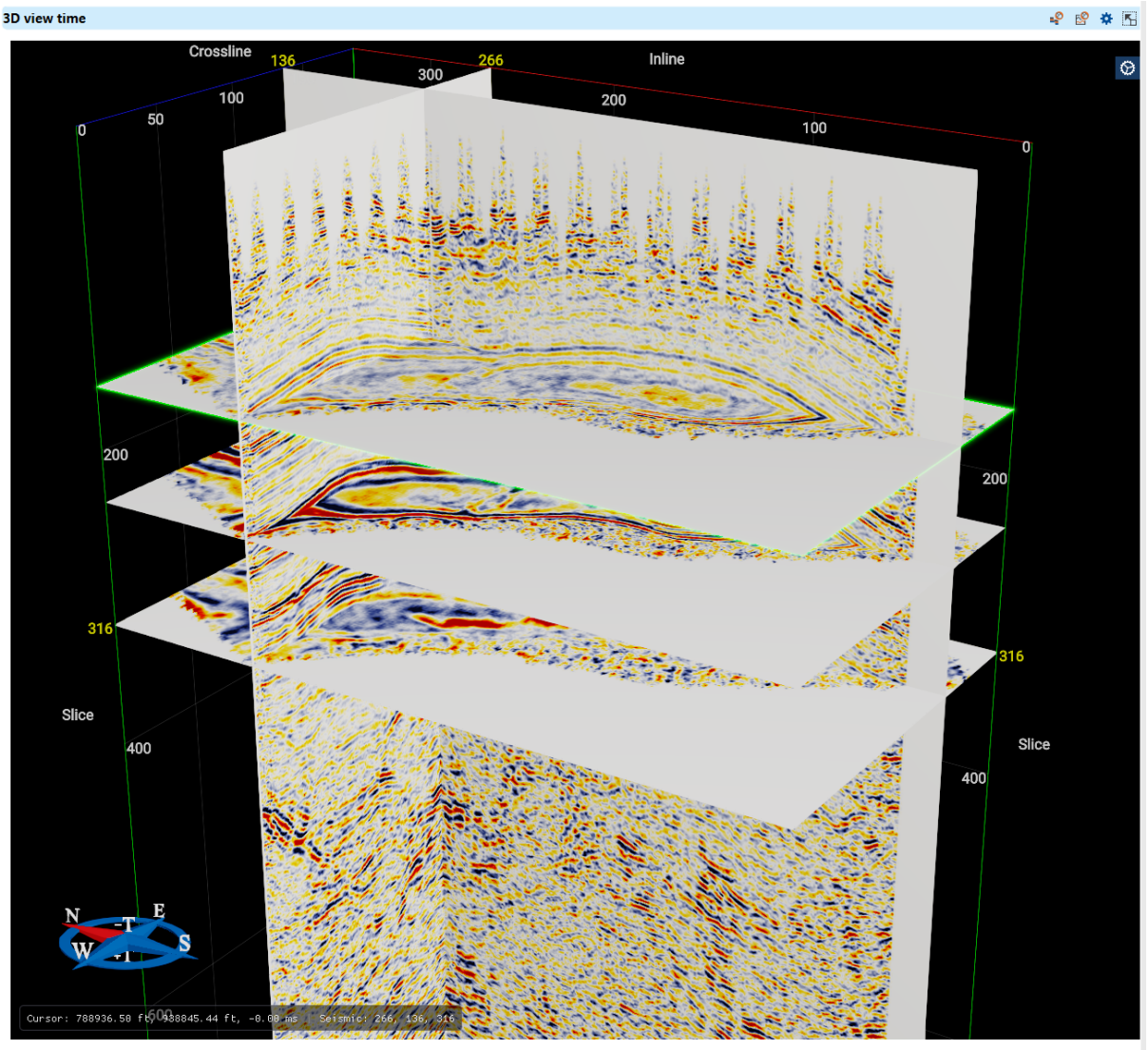
For removing them click Delete.

© 2026 Geomage 2003 LTD



For additional slices click **S** key.

© 2026 Geomage 2003 LTD



Using the **Arbitrary Line** Tool in g-Space

Arbitrary Line is particularly useful in the following scenarios:

- **Comparing 2D and 3D Data:** This tool is essential for comparing and assessing the quality of different seismic surveys. By creating custom lines that traverse both 2D profiles and the 3D seismic volume, users can directly compare the resolution and continuity of seismic reflections, leading to a better understanding of the subsurface. A detailed description is available in the 2D data 589 chapter.

© 2026 Geomage 2003 LTD

- **Correlating Wells:** The arbitrary line is helpful for correlating wells, especially when you need to understand the structural and lithofacies features of the interwell space. By drawing a line that connects multiple wells, you can visualize how stratigraphic horizons and structural features change between wells.
- **Tying Directional Wells:** When dealing with directional or deviated wells, the arbitrary line allows you to create cross-sections that follow the wellbore trajectory. This aids in accurately tying the well data to seismic data, ensuring a more precise integration of well and seismic interpretations.
- **Analyzing Geological Structure:** For fields with complex geological features, such as the anticline fold in the Demo project, the arbitrary line tool is particularly useful for analyzing geological structures both along and across the fold axis. Given that the fold has a curved structure relative to the geometry of the seismic survey, arbitrary lines enable a more flexible and accurate examination of the fold's features, which might not align with standard seismic lines.

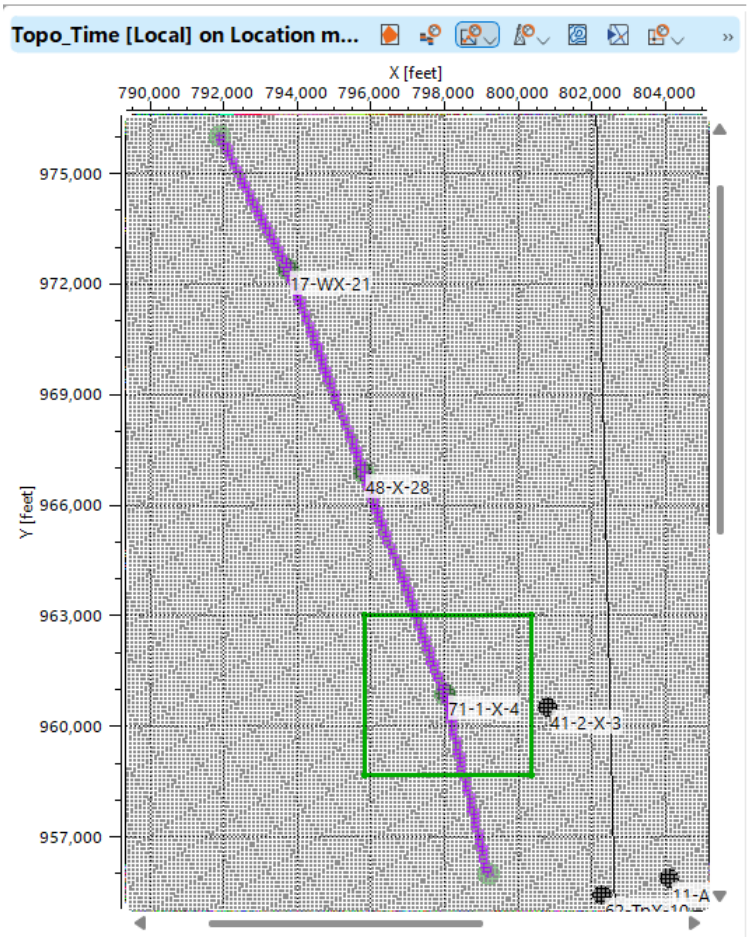
Creating **Arbitrary line** using wells location.

In the **Data Manager**, create a new **Arbitrary line** and name it.

Visualize the wells and 3D seismic in the Map window. From the top menu, select the **3D arbitrary line** tool and draw the line through the well locations. Depending on your objectives, you may find it useful to visualize either the wellheads or the projection of the entire well trajectory.

To remove points from the arbitrary line, hold down the right mouse button and select the unwanted point(s) (highlighted by a green polygon).

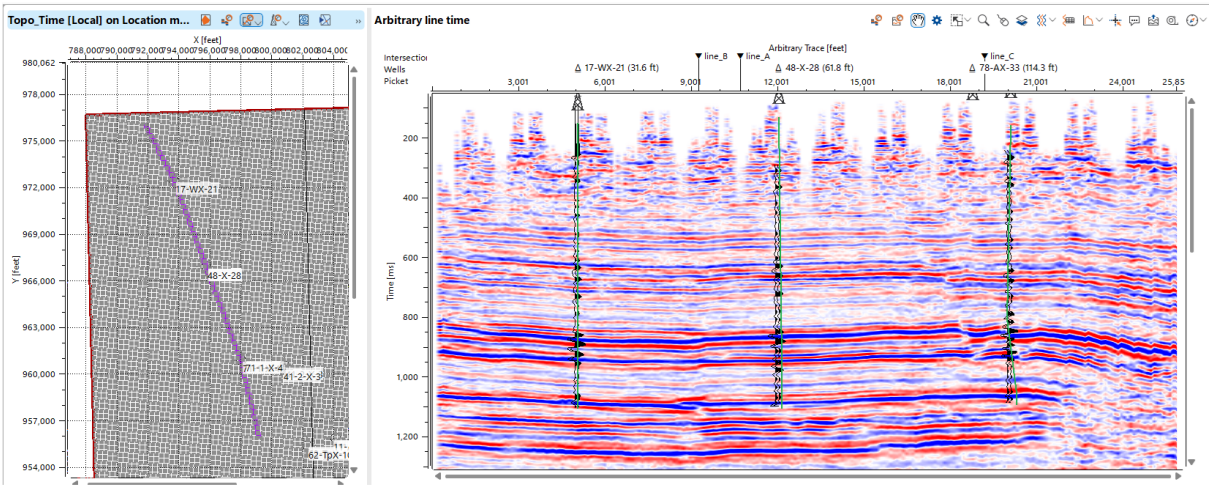
© 2026 Geomage 2003 LTD



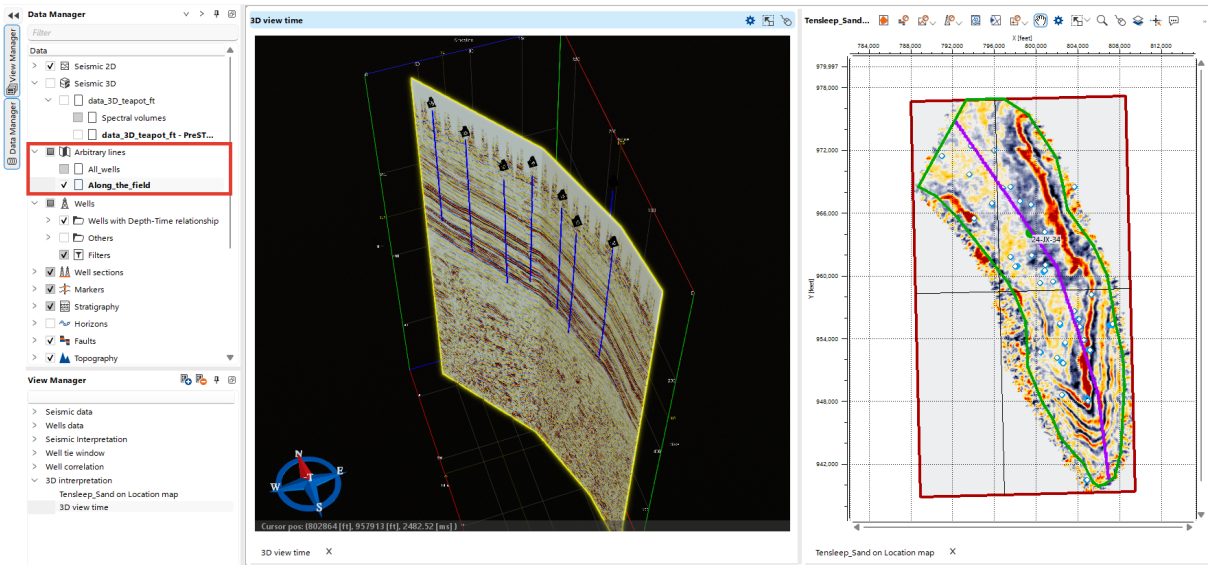
On the created arbitrary line, you can visualize well data, including well trajectories or vertical projections of the wellheads. Additionally, it's possible to display logging curves and synthetic traces for further analysis.

To display wells in the time domain, the user must have a checkshot.

© 2026 Geomage 2003 LTD

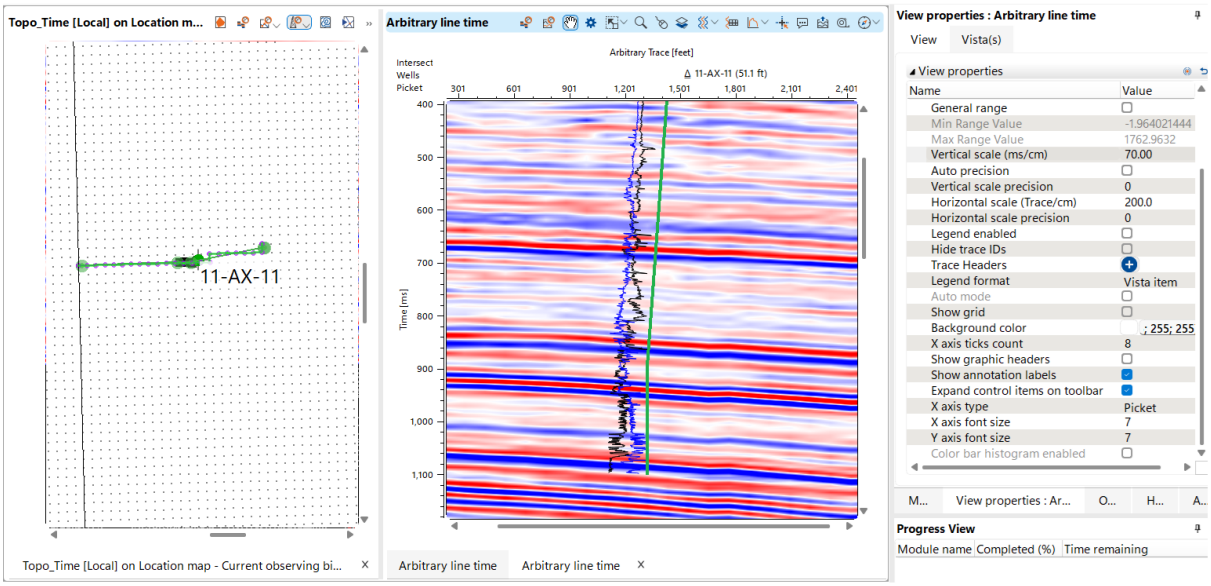


To display an arbitrary line in 3D View, open the Time/Depth 3D View. In the Data Manager, select the arbitrary line(s) you wish to visualize.



To display a seismic section along a deviated well, it is useful to create an arbitrary line with such a geometry that the trajectory lies within a single plane.

© 2026 Geomage 2003 LTD



g-Space supports the creation of multi-panel displays where you can visualize different types of views (e.g., inline, crossline, time slice) simultaneously. This setup provides a more comprehensive interpretation workspace.

9.5.2 Horizon picking

Seismic horizon correlation is aimed at mapping continuous reflectors that represent geological horizons across a seismic dataset. The process relies on several core principles:

Waveform Continuity, Stratigraphic Context, Well Data Integration, Spatial Consistency.

These principles underpin the practical steps for seismic horizon correlation in g-Space, which includes Well tie 567, Horizon picking, and Mapping 724.

Notes

Methodological Approaches

1. Phase correlation focuses on tracing reflectors by maintaining the continuity of specific phases (peaks, troughs, or zero crossings).

This method is effective in areas with clear, consistent reflectors. It involves:

- *Picking a phase (e.g., peak or trough) and tracing it across the dataset.*

© 2026 Geomage 2003 LTD

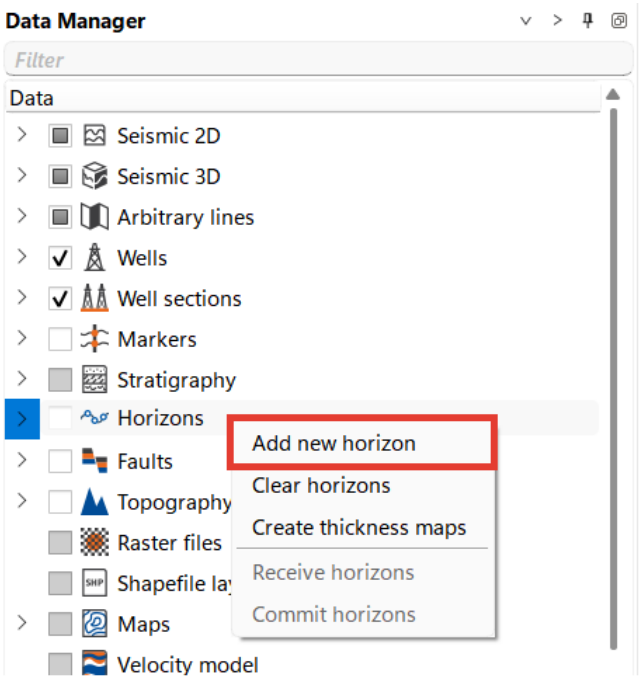
- *Ensuring that the selected phase remains consistent across traces, using g-Space tools to automatically track reflectors where possible.*

2. Seismostratigraphy is the process of identifying depositional sequences and stratigraphic surfaces from seismic reflection patterns. This method is often used to:

- *Identify key stratigraphic boundaries (e.g., unconformities or maximum flooding surfaces).*
- *Correlate horizons based on depositional features such as onlaps, downlaps, or erosional surfaces.*

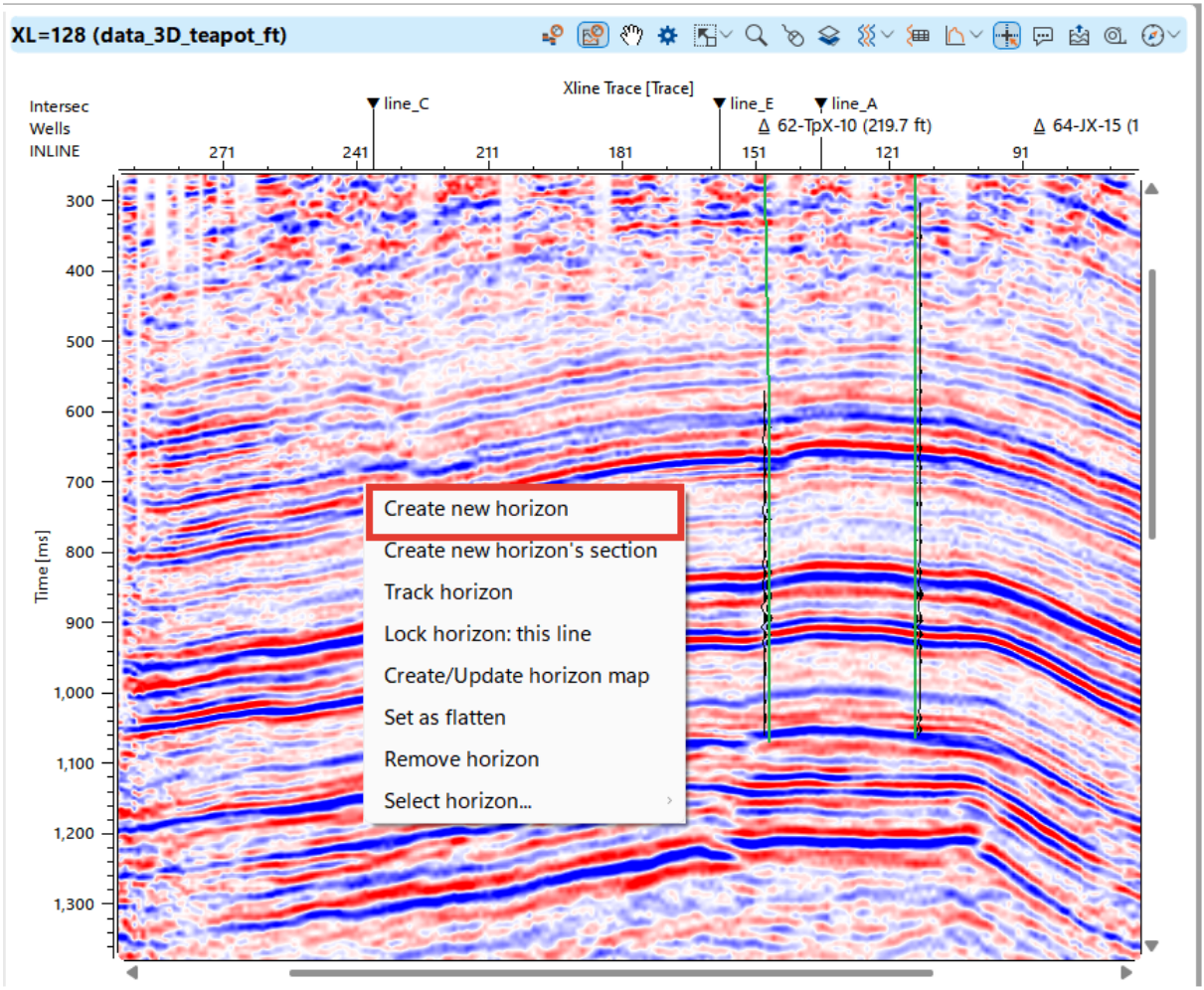
Begin picking horizons at well locations where the reflectors have been tied to known formations using the Well tie 93 functionaity.

To create a new horizon go to Data manager 124 and press Right Click Mouse on the Horizon icon. Choose **Add new horizon**



or on Seismic view in tool bar press Horizon picks button and press **Alt + Right Click Mouse** button on seismic view and choose **Create new horizon**

© 2026 Geomage 2003 LTD



Let's set up a workspace for horizon picking. In this example, a map is displayed showing a seismic volume, inline and crossline sections, wells, as well as inline and crossline views with linked wells, synthetic traces, and markers. The Interpretation bar 96 provides the picking settings used in this workflow.

Subscribe to our channel and watch the video on how to pick horizons in g-Space on YouTube

Click on **Horizon Pick** (highlighted in red) in the top panel of the view to begin picking horizons.

The same **Horizon picks** tool is available outside a time section: on depth **Inline**, **Crossline** and **Arbitrary line** sections (see Seismic view 240 for opening a view in the depth domain) and in the 3D **Time** and 3D **Depth** views 244. All of them also carry the **Unlocked horizon picks** and **Tracking horizon picks** items, so manual and tracking picks stay visible while you work. In

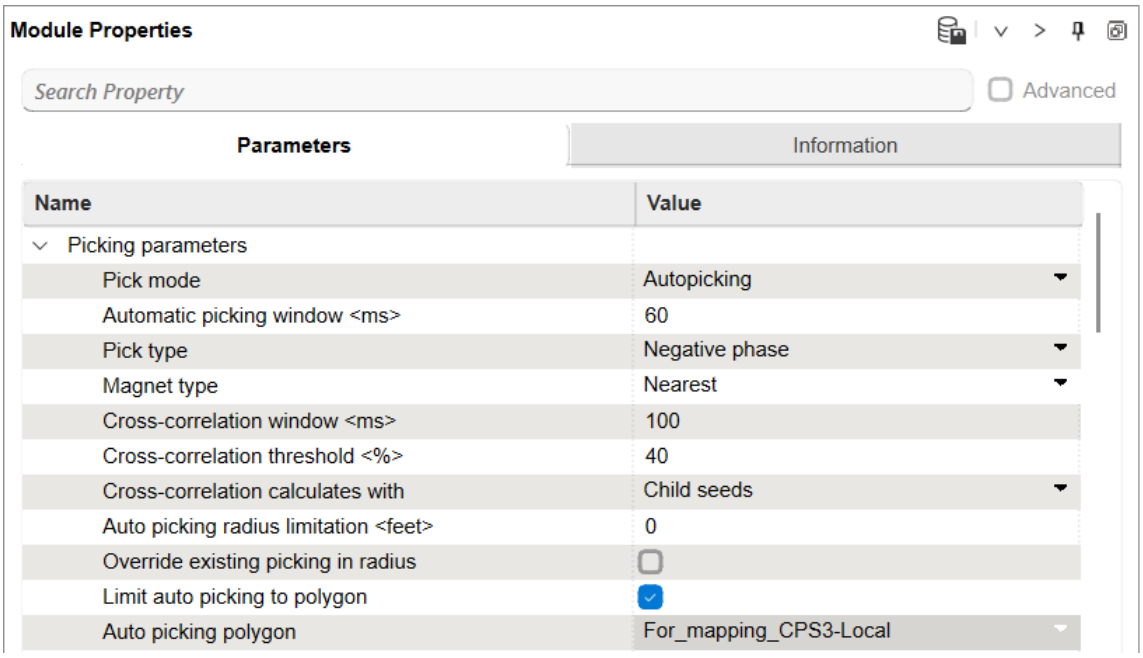
© 2026 Geomage 2003 LTD

a 3D view you pick by clicking a section shown in the scene - a 2D line, an arbitrary line, an inline or crossline section, or an inline or crossline face of a seismic volume - and the picks go to the current horizon.

Only your own horizons can be edited. When the horizon and the view are in different domains - a time horizon picked on a depth view, or a depth horizon picked on a time view - g-Space converts the pick through the current velocity model 658, so select one first; without it the pick is not added.

Set the picking behavior in the **Picking** group on the Interpretation bar 96. **Pick mode** is set to **Autopicking** by default, so change it there first if you want to pick manually. The mode-specific workflows are described below.

Autopicking can also be restricted to one area of the survey. Open the Module Properties 197 panel and expand the **Picking parameters** group. While **Pick mode** is **Autopicking**, this group also offers the **Limit auto picking to polygon** checkbox (cleared by default) and the **Auto picking polygon** drop-down (empty by default). These two settings stay in **Module Properties** and are not part of the **Picking** group on the bar.



The drop-down lists every polygon 167 of the project as **<polygon name>-<user>**, and its empty first entry means no restriction. The list is filled when a project is opened and refreshes automatically when polygons are added, renamed, deleted or imported, so a polygon stored with the project can be selected straight away.

© 2026 Geomage 2003 LTD

Select the checkbox and choose a polygon to autopick only the bins inside it: the spread stops at the polygon boundary, and bins outside it are neither picked nor overwritten. The polygon limit works together with **Auto picking radius limitation**, so when both are set a bin has to satisfy both. Clear the checkbox or leave the polygon empty to remove the restriction. The limit applies to **Autopicking** only: in **Tracking**, **Interpolation**, **Continuous** and **Autopicking -- Multi-Horizon** mode the two settings are hidden and have no effect.

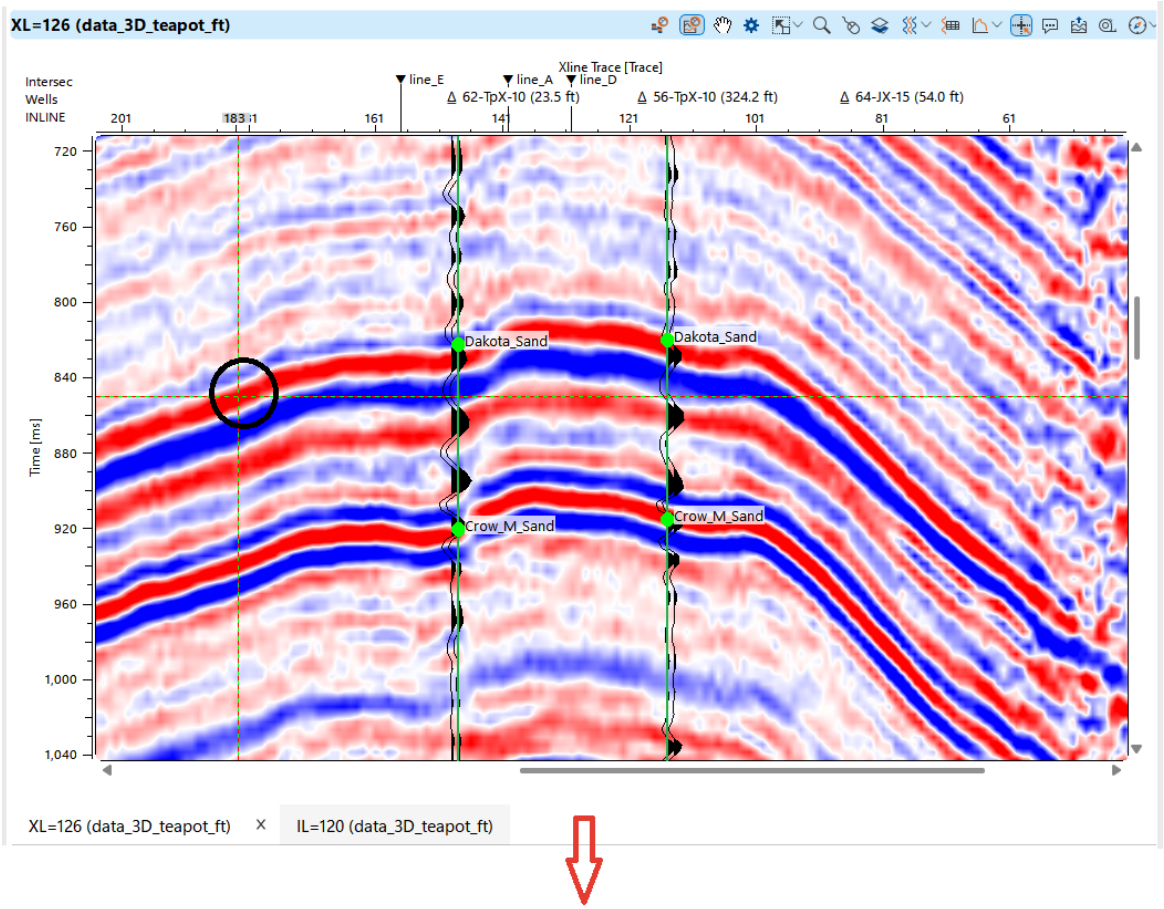
The **Pick mode** list offers **Tracking**, **Interpolation**, **Continuous**, **Autopicking** and **Autopicking -- Multi-Horizon**, the last of which autopicks several horizons in one pass. The **Tracking**, **Interpolation** and **Continuous** workflows are described below:

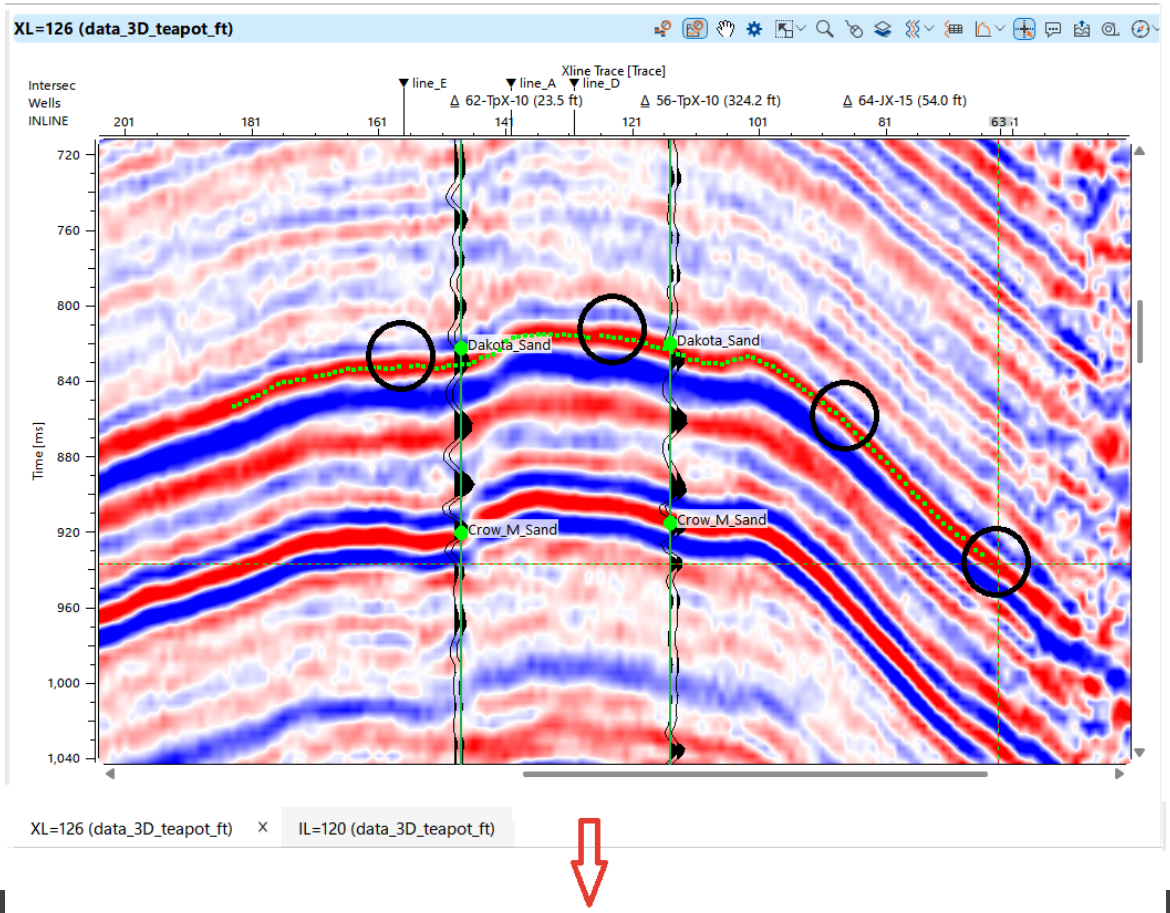
1. **Tracking**: Automatically follows seismic events across the section based on the selected pick phase. Click the left mouse button, in the example the click points are highlighted with black circles, to finish, click the left mouse button twice

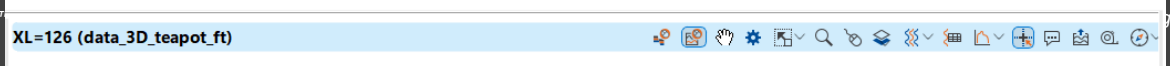
Notes

In tracking mode, the software automatically follows a seismic event based on amplitude and phase consistency. This is ideal when horizons are well-defined and exhibit minimal interruptions or distortions, such as in clear stratigraphic layers. It reduces the amount of manual picking and ensures consistent interpretation over large sections.

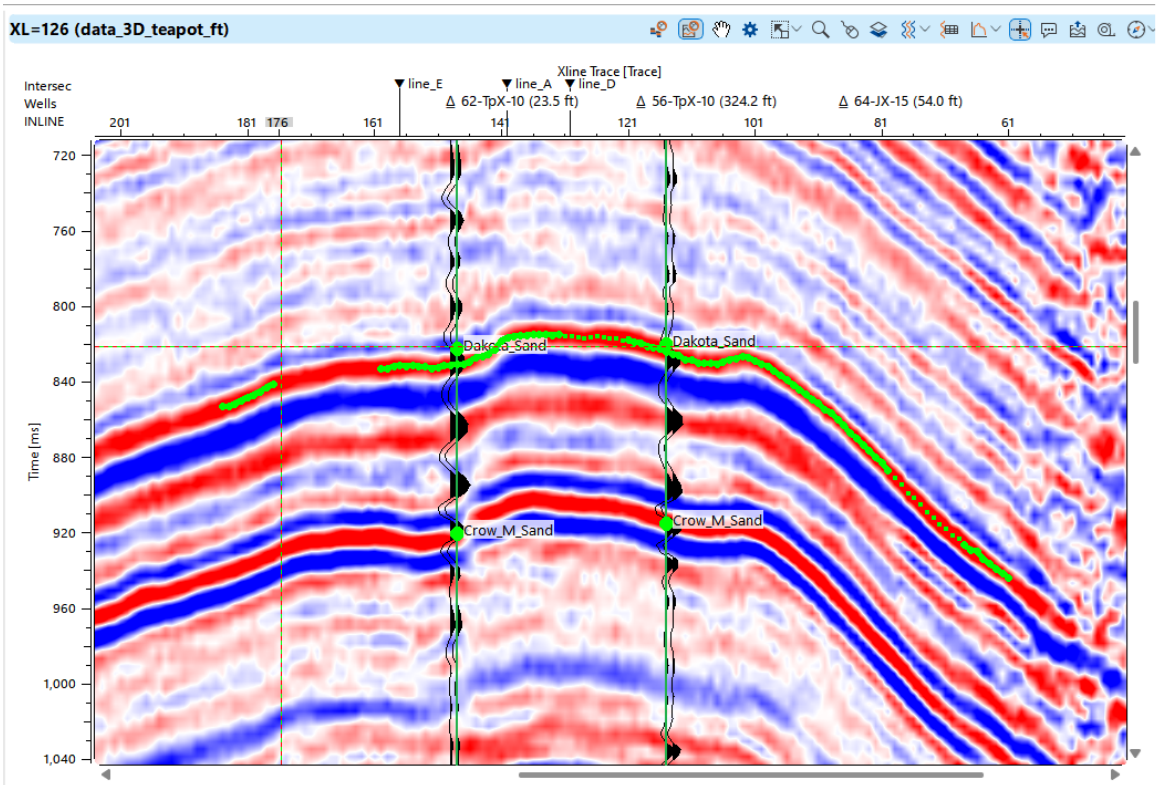
© 2026 Geomage 2003 LTD


XL=126 (data_3D_teapot_ft)
Intersec
Wells
INLINE
Xline Trace [Trace]
line_E
line_A
line_D
Δ 62-TpX-10 (23.5 ft)
Δ 56-TpX-10 (324.2 ft)
Δ 64-JX-15 (54.0 ft)
Time [ms]
Dakota_Sand
Crow_M_Sand
XL=126 (data_3D_teapot_ft)
IL=120 (data_3D_teapot_ft)


XL=126 (data_3D_teapot_ft)
Intersec
Wells
INLINE
Xline Trace [Trace]
line_E
line_A
line_D
Δ 62-TpX-10 (23.5 ft)
Δ 56-TpX-10 (324.2 ft)
Δ 64-JX-15 (54.0 ft)
Time [ms]
Dakota_Sand
Crow_M_Sand
XL=126 (data_3D_teapot_ft)
IL=120 (data_3D_teapot_ft)


XL=126 (data_3D_teapot_ft)

To delete a segment of interpretation, press and hold the right mouse button and select the area to delete. You can also immediately proceed to editing and put new points on these segments.



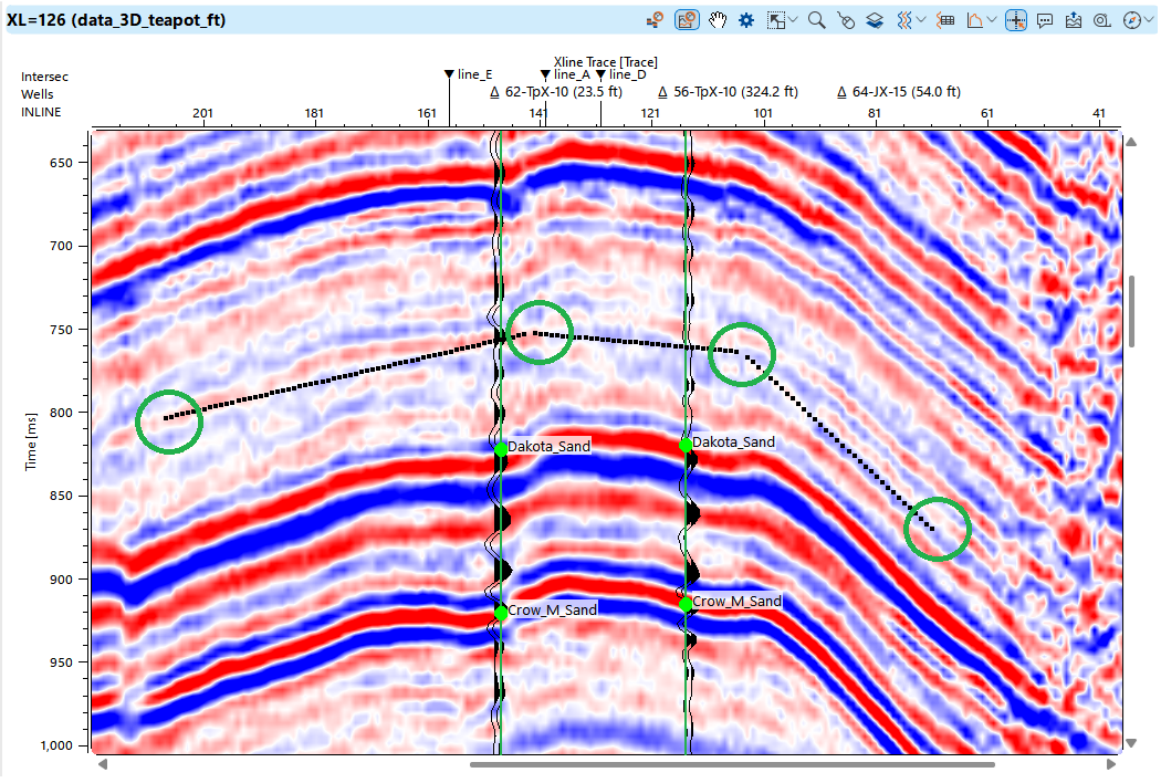
In **Tracking mode**, when moving to another section, right-click the seismic view and choose **Track horizon**. g-Space automatically correlates the horizon on the newly opened section.

2. **Interpolation**: Picks points at intervals and interpolates between them to define the horizon.

Notes

Interpolation mode is useful when seismic horizons are partially obscured, interrupted by faults, or have variable continuity. The user selects key points along the horizon, and the software interpolates between these points to create a complete pick. This approach balances manual input with automatic interpretation, allowing for more flexibility in challenging areas with less-defined reflectors.

© 2026 Geomage 2003 LTD

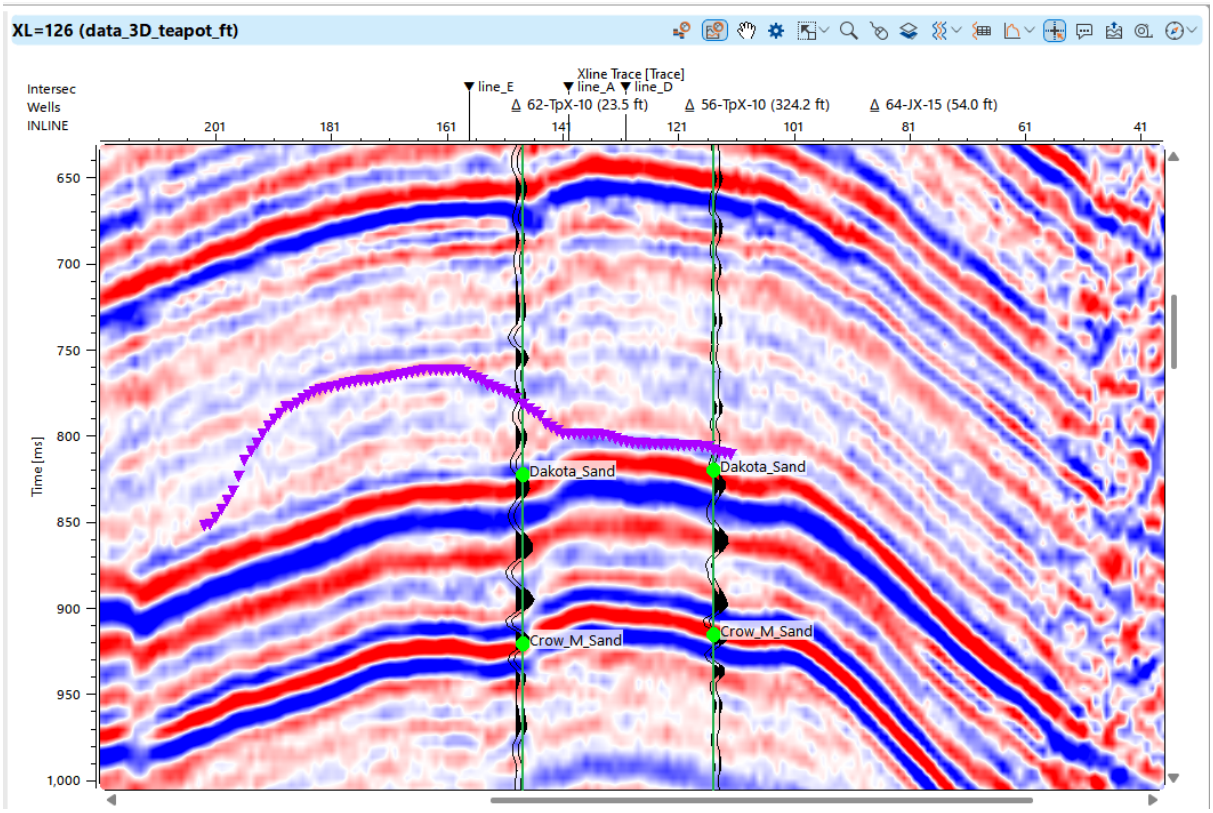


3. **Continuous**: Enables continuous manual picking across the section, allowing the user to define the horizon in real-time. The process looks like drawing on a seismic image.

Notes

Continuous mode is fully manual and allows the interpreter to continuously pick the horizon across the section. This mode is helpful when there are significant geological complexities, such as in areas with faulting, folding, or high noise levels, where automated tracking or interpolation might not perform accurately. The interpreter retains full control over the horizon pick, ensuring precision in difficult regions.

© 2026 Geomage 2003 LTD



Hot keys for seismic section views:

Up and **Down** move the active section up and down in the set scale

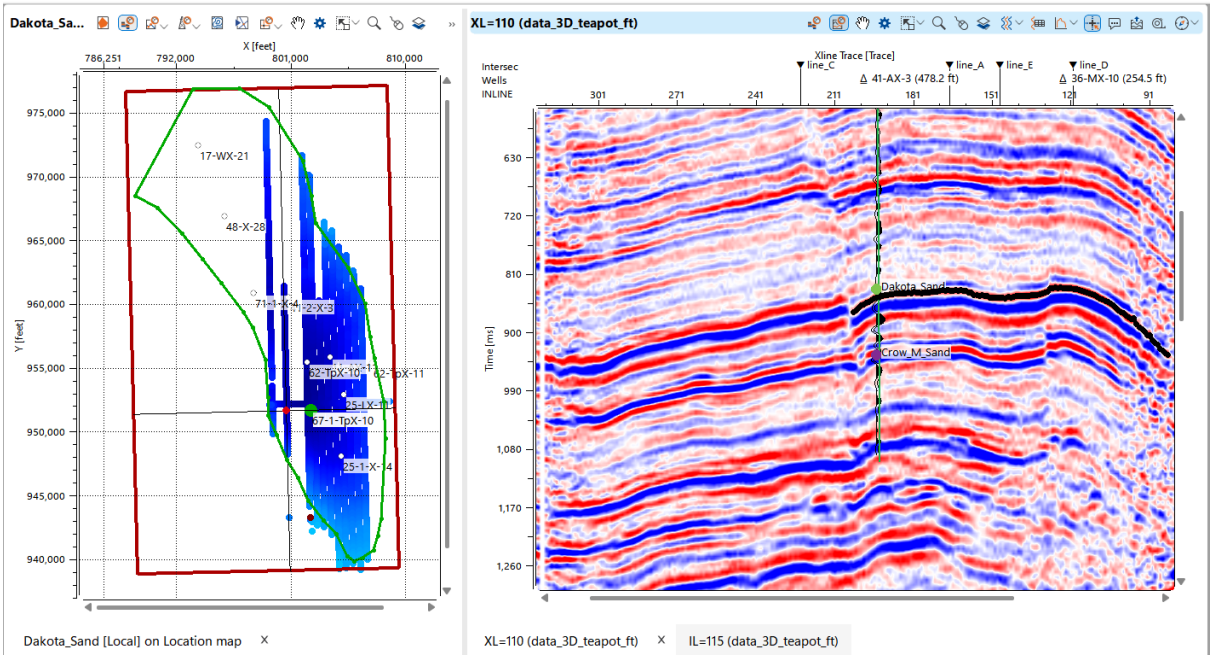
Ctrl+Left and **Ctrl+Right** move crosslines with a set step

Ctrl+Up and **Ctrl+Down** move inlines with a set step

Or **Page Up** and **Page Down** buttons move both inlines and crosslines in the active view

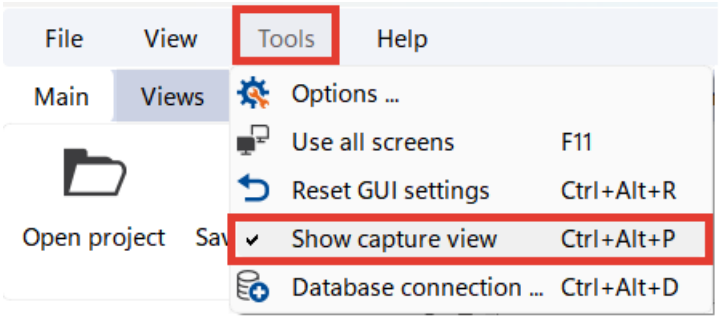
During the horizon picking process, the horizon surface is updated in real-time within the map view.

© 2026 Geomage 2003 LTD



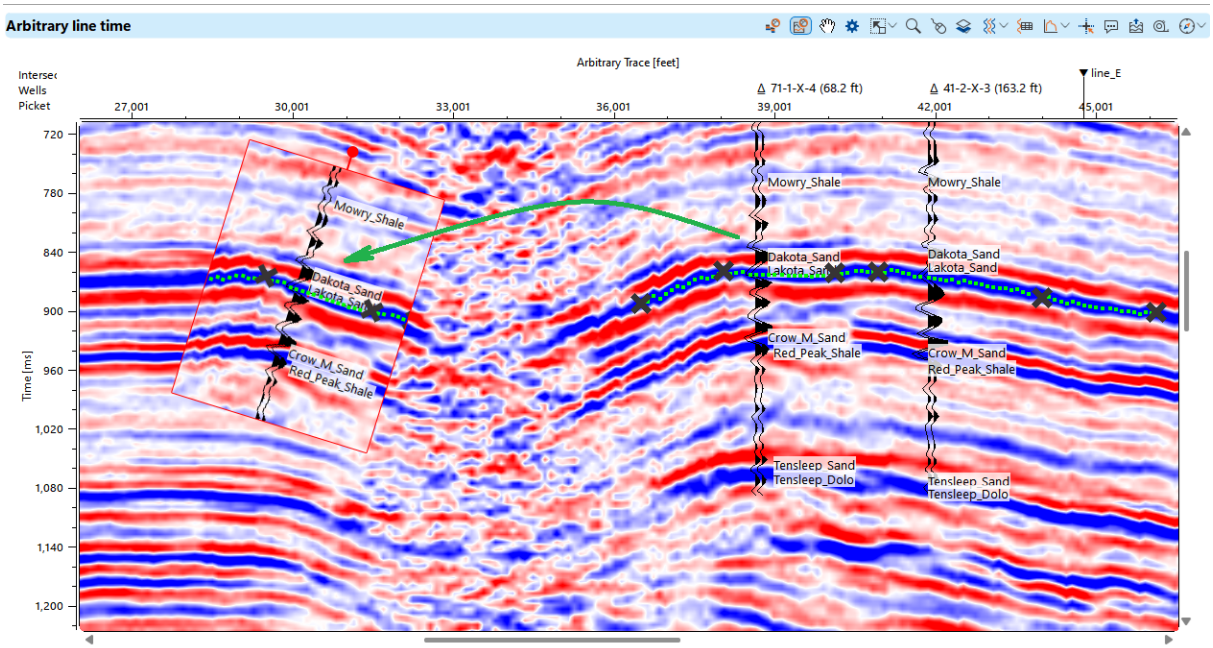
When interpreting seismic volume areas remote from wells, across fault blocks, or around geological features that disrupt reflecting horizons, the ghosted view tool in g-Space is particularly useful. This tool allows the user to take a screenshot of a specific interpretation, drag it to a new location, and overlay it on the current view, simplifying the horizon picking process.

Additionally, the tool enables rotation of the screenshot, making it adaptable to various structural alignments. This method is particularly effective in complex tectonic settings, ensuring more accurate seismic horizon correlation.



Open drop-down menu in Tools in Main bar 35 and choose Show capture view or use hot keys **Ctrl + Alt + P**

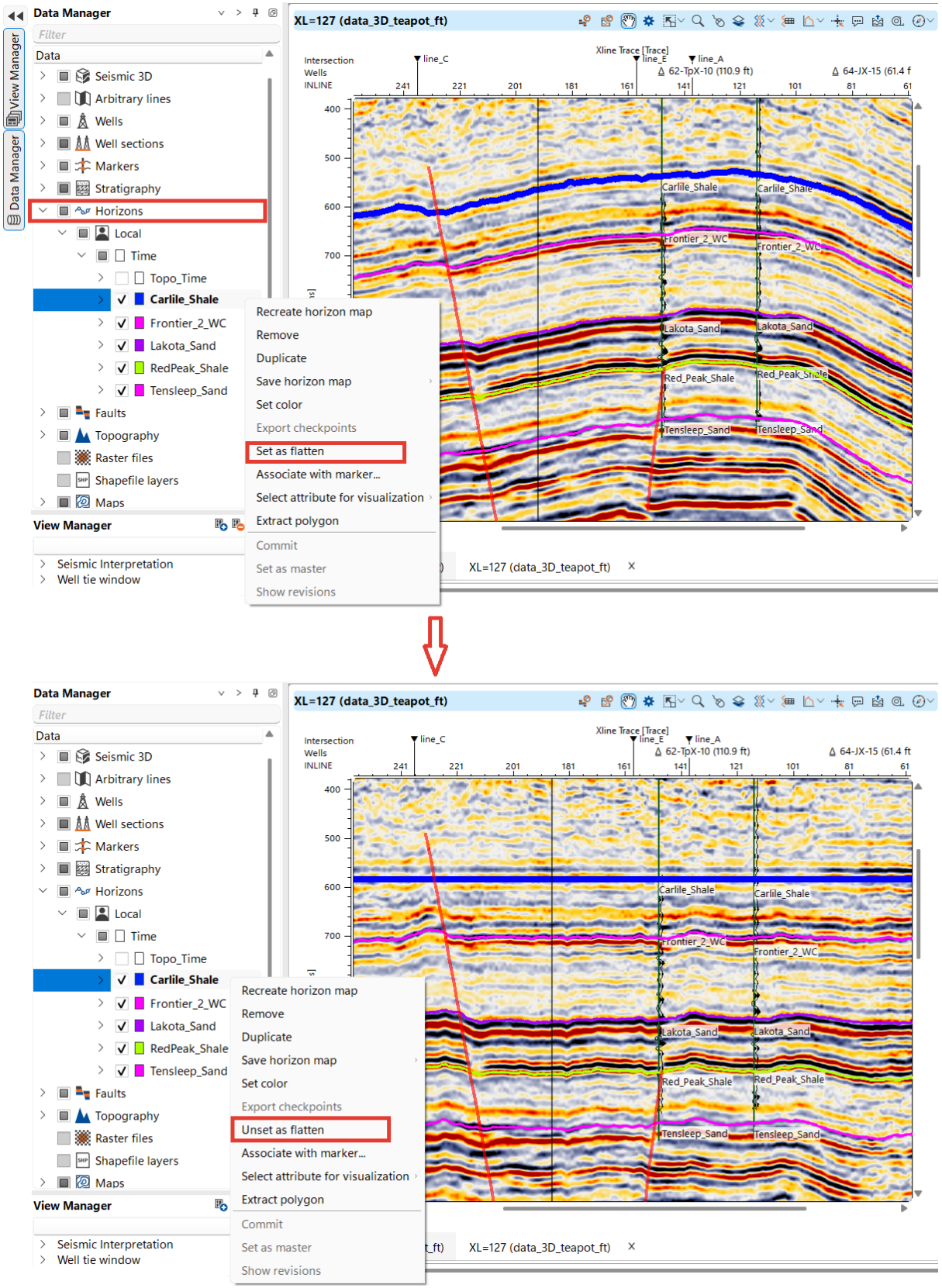
© 2026 Geomage 2003 LTD



The **Flatten Horizon** tool can be very useful to effectively remove structural complexities and focus on stratigraphy.

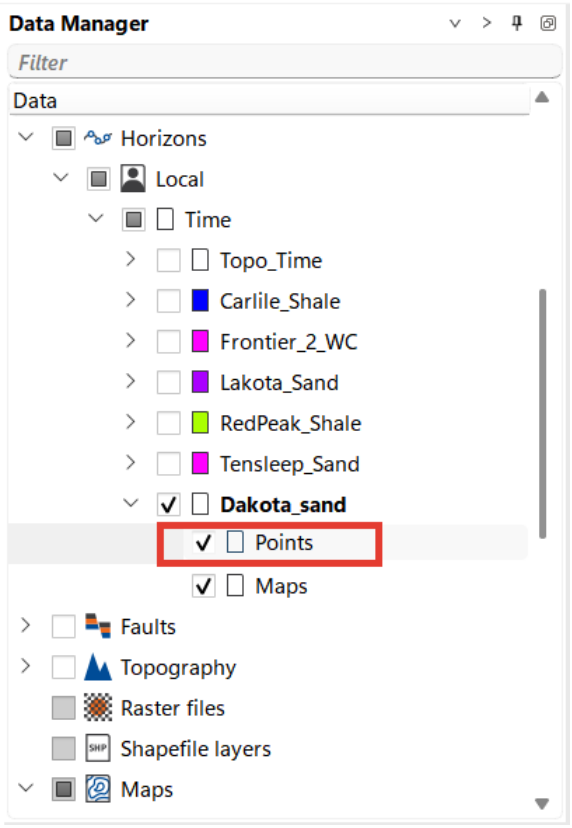
To flatten a seismic section to a specific horizon, right-click on the horizon name and select **Set as flatten** from the drop-down menu. The seismic section will then be flattened to the selected horizon. To return to the normal view, repeat the steps and choose **Unset as flatten**.

© 2026 Geomage 2003 LTD



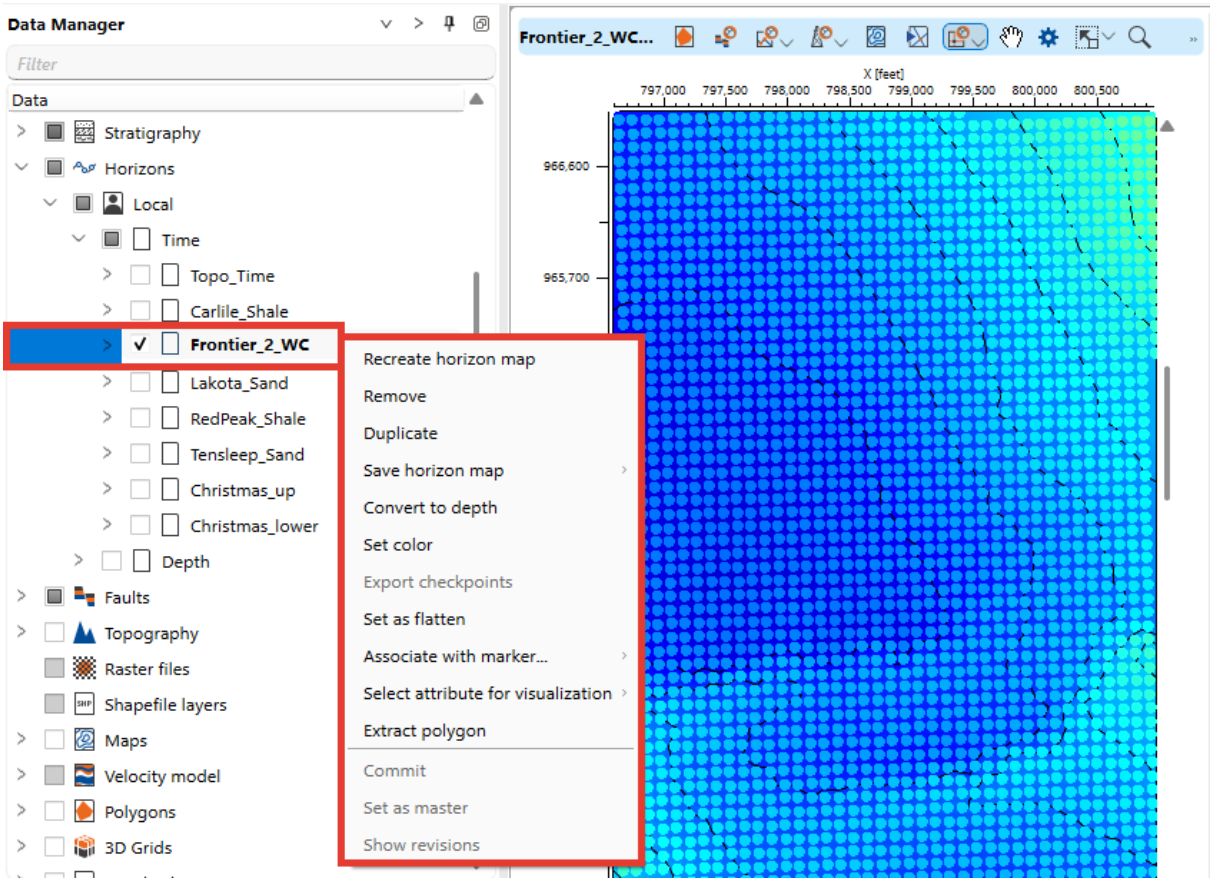
© 2026 Geomage 2003 LTD

The outcome of the interpretation consists of points, each associated with a specific set of attributes, Time in our case.



There is a drop down menu in Data manager for Horizons where user can create maps, flatten horizons, set colors, etc. At any point user can rebuild the map of the selected horizon.

© 2026 Geomage 2003 LTD



For **3D visualization**, g-Space offers a feature that allows **time-domain data to be displayed within the 3D depth view** using the current velocity model.

This is especially useful during interpretation, as it enables reviewing results in time without requiring full data conversion to the depth domain

9.5.3 Fault picking

Fault picking is aimed at identifying discontinuities or breaks in subsurface reflectors that indicate faults. The process in g-Space follows core principles similar to those used in Horizon picking 614 but with an emphasis on identifying and mapping structural breaks.

Notes

© 2026 Geomage 2003 LTD

Methodological Approaches

1. *Discontinuity Identification Fault picking focuses on identifying breaks or offsets in seismic reflectors. The key steps include:*
    - *Locating regions where reflectors are abruptly truncated, offset, or disrupted.*
    - *Using g-Space tools, such as attribute maps, to highlight areas of high discontinuity or variance in reflector continuity.*
2. *Structural Interpretation Faults often follow particular structural patterns, which can help guide the picking process. This involves:*
    - *Recognizing common fault geometries (e.g., normal, reverse, or strike-slip faults).*
    - *Identifying associated structures like rollover anticlines or drag folds that accompany fault systems.*
3. *Seismostratigraphic Context Faults often correlate with stratigraphic markers. Integrating seismostratigraphic information can help refine fault interpretation by linking faults to stratigraphic units, sedimentary processes, and depositional environments.*

The Data Manager 124 in g-Space organizes fault data into a structured hierarchy that allows users to work within either the **Time** or **Depth** domain, depending on the seismic data and the interpretation workflow.
The Local folder is active by default for multi-user mode and the results of interpretation are stored in your local project.

Free Sticks:
Faults can be visualized and manipulated as Free Sticks, representing vertical fault traces along seismic sections. This view allows users to pick fault segments and analyze them in a stick-based representation, which is useful for initial interpretations and quality control.

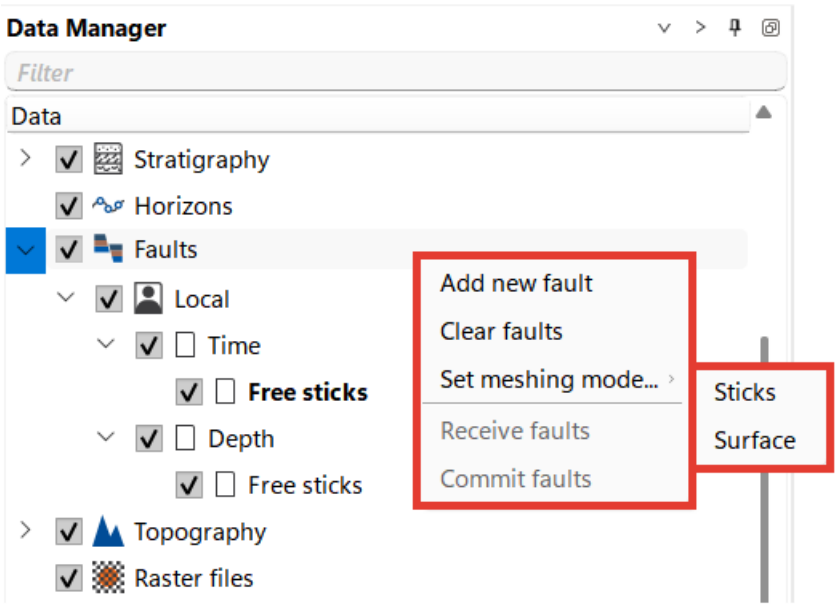
Surface Mode:
Users can switch from the stick representation to Surface Mode, which converts the fault sticks into continuous fault surfaces. This mode is helpful for visualizing the complete geometry of a fault plane, providing a more holistic view of the fault structure within the seismic volume.

You can define faults in the **Map View**, **3D View**, **2D View**, **Inline** and **Crossline** Views as well as in the **Arbitrary line View**.

Picking is not limited to the time domain: depth **Inline**, **Crossline** and **Arbitrary line** sections offer the same **Fault picks** tool and the same **Current fault** item as their time counterparts. Each stick keeps its own **Time** or **Depth** domain, and a click made on a section in the other domain is converted through the current velocity model 658, or through the project **V0** value when no velocity model is selected, so a time fault can be extended from a depth section and vice versa.

© 2026 Geomage 2003 LTD

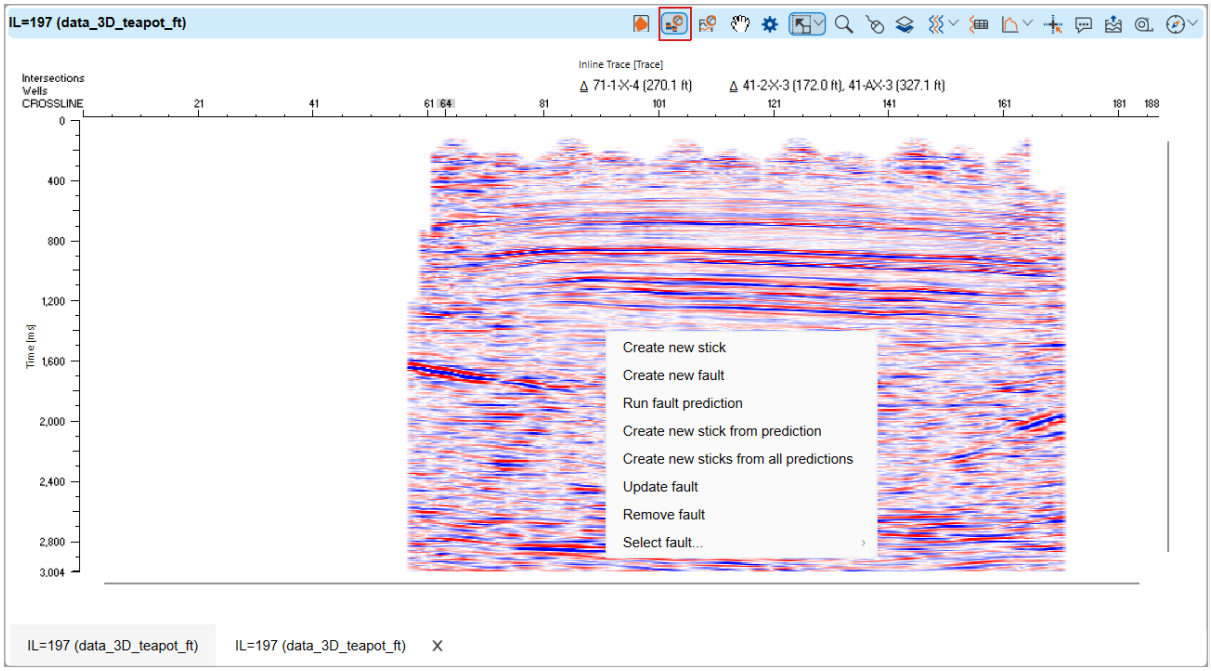
g-Space also provides options to Import faults 441.



To start fault picking, create a new fault in the **Data Manager** or directly on the seismic section by pressing **Alt + Right Click Mouse** with the **Fault Picks** mode enabled (highlighted in red)

On a time section this menu also offers **Run fault prediction**, **Create new stick from prediction** and **Create new sticks from all predictions**, described in Fault prediction 640. These three commands are not offered on a depth section.

© 2026 Geomage 2003 LTD

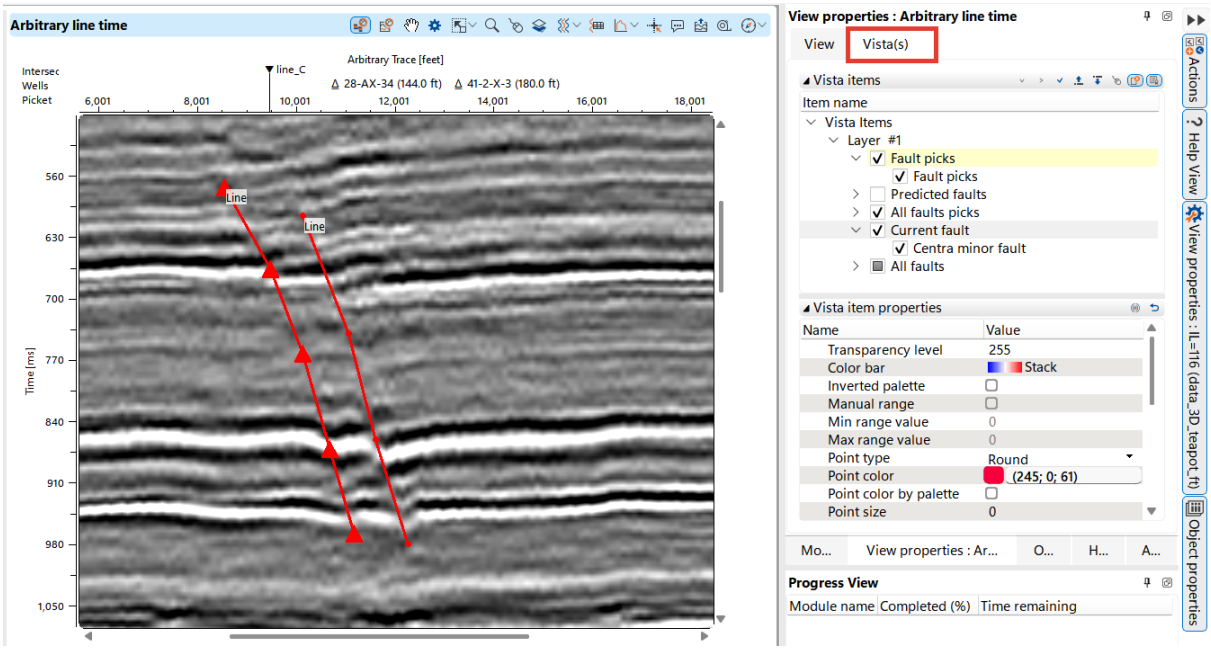


Users have access to various settings for visualizing faults in the Visual Settings 206 panel.

Make the object you want to customize active in the Data Manager 124, it will appear in the **Object(s)** tab and visual settings will be available for each element at the bottom of the right area at the **Object item properties**.

In the image below, a color palette is configured specifically for the seismic section, while the visualization of already picked faults and the current fault is managed separately.

© 2026 Geomage 2003 LTD



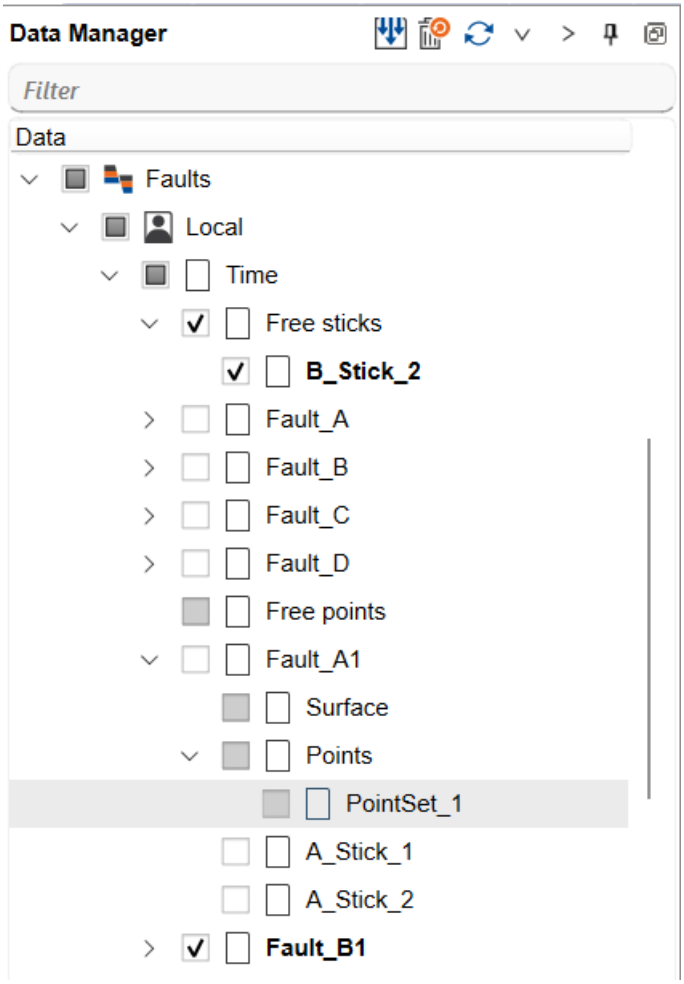
,

During the fault picking process, the user creates sticks within the folder of a specific fault. If a stick is mistakenly assigned to the wrong fault, the interpreter can easily correct this by selecting the stick with the left mouse button and dragging it to the correct fault folder. In cases where it is unclear which fault a particular stick belongs to during interpretation, the user can temporarily store it in the **Free Sticks** folder for later assignment.

A fault also stores point sets in its **Points** folder. To add one, right-click **Points** under a fault you can edit and choose **Create empty point set**; the new set appears under **Points** as **PointSet_1**, **PointSet_2** and so on. Right-click a point set and choose **Rename** to open the **Set name of the point set** dialog and enter a new **PointSet name**, or choose **Remove** to delete the set. Both commands are offered only for faults you can edit. Point sets are stored with the fault and saved with the project.

Each fault also has a **Surface** node that holds its triangulated fault surface. Select the **Surface** check box to draw that surface in the 3D view 244; it stays in step with the matching item under **Fault 3D** in the Visual Settings 206 panel. The sticks are controlled by the fault's own check box, so the surface and the sticks can be shown separately or together. To write the displayed surface to a GOCAD TSurf file, right-click **Surface** and choose **Export to GOCAD TSurf**; see GOCAD Data Formats 519.

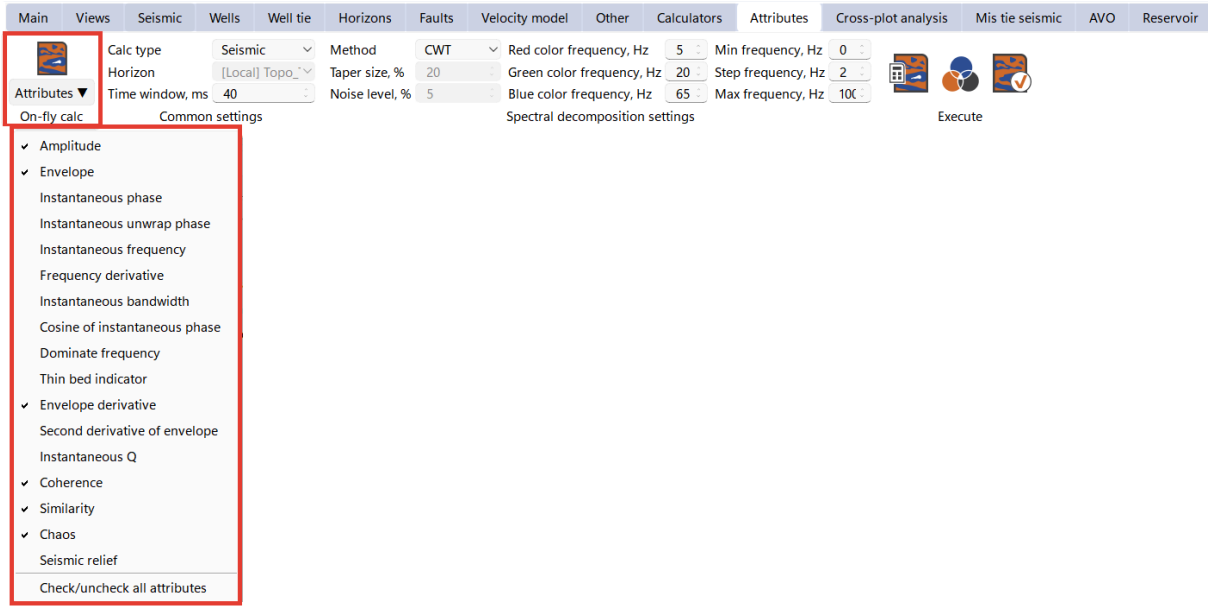
© 2026 Geomage 2003 LTD



It is a common practice to use seismic attributes to enhance fault identification. While there is an entire chapter dedicated to Attribute analysis 680, here we will focus specifically on the **On fly attributes calculations** to aid in fault picking.

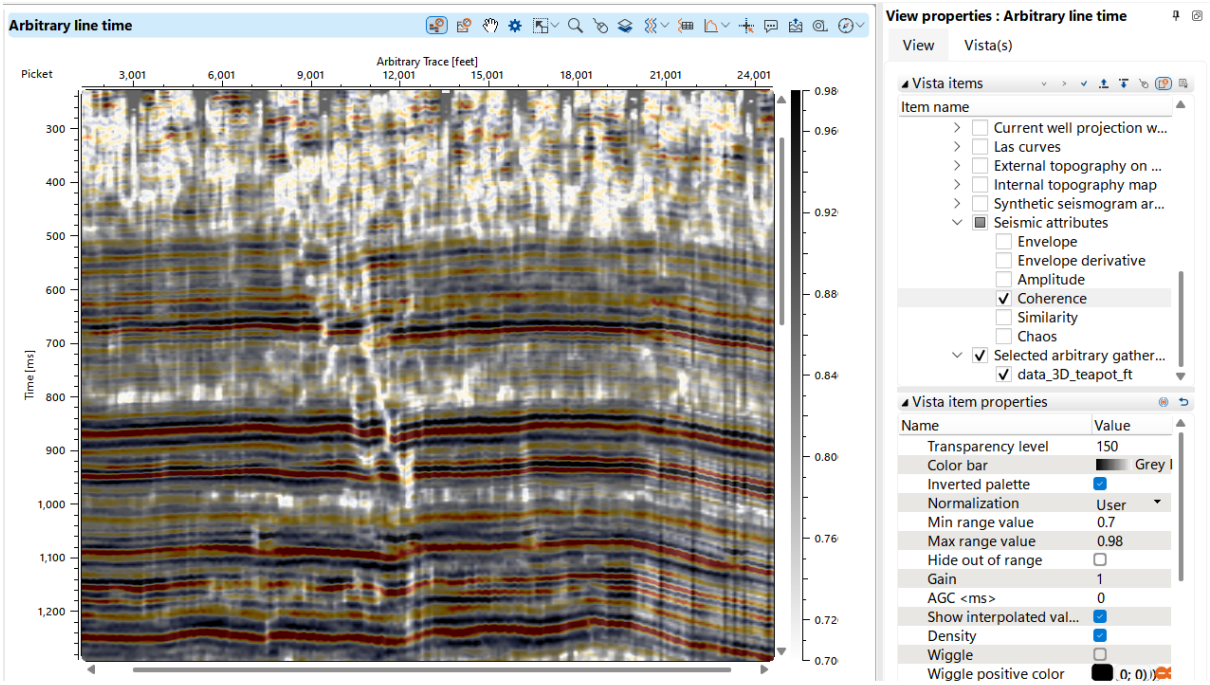
In the Attributes Bar 105, select **On fly calculation** and select the attribute you need, it will instantly appear on the section you have open.

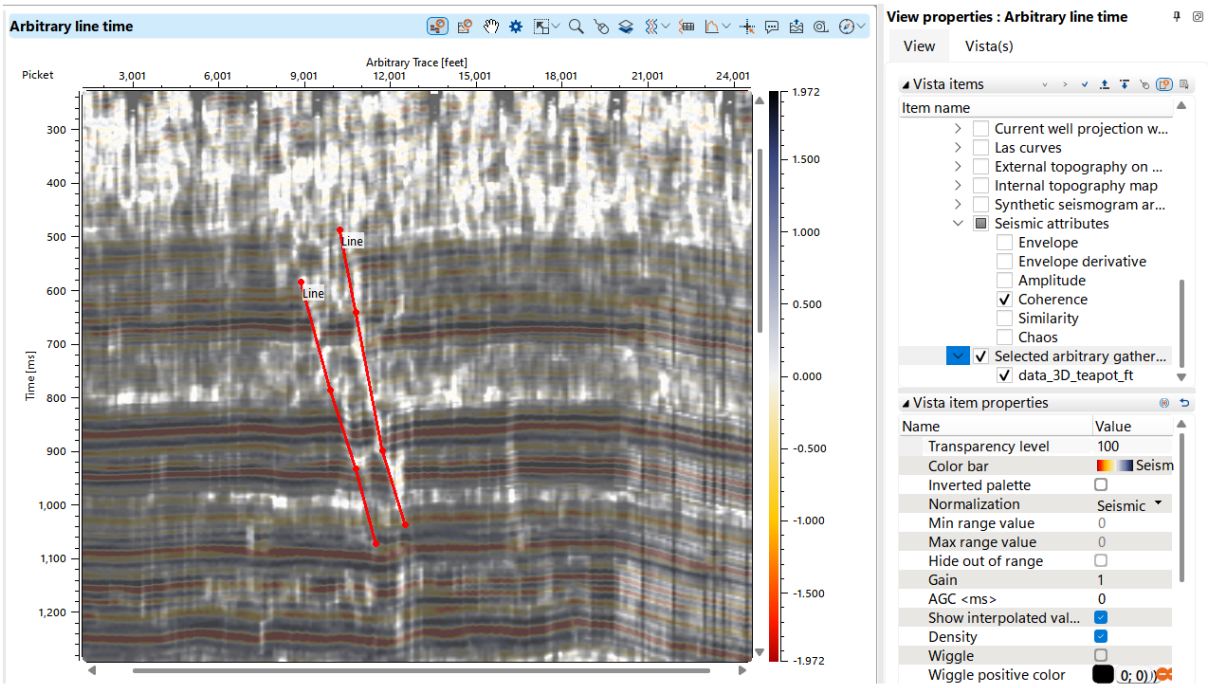
© 2026 Geomage 2003 LTD



Set up the palette for maximum efficiency and perform a fault pick.

© 2026 Geomage 2003 LTD



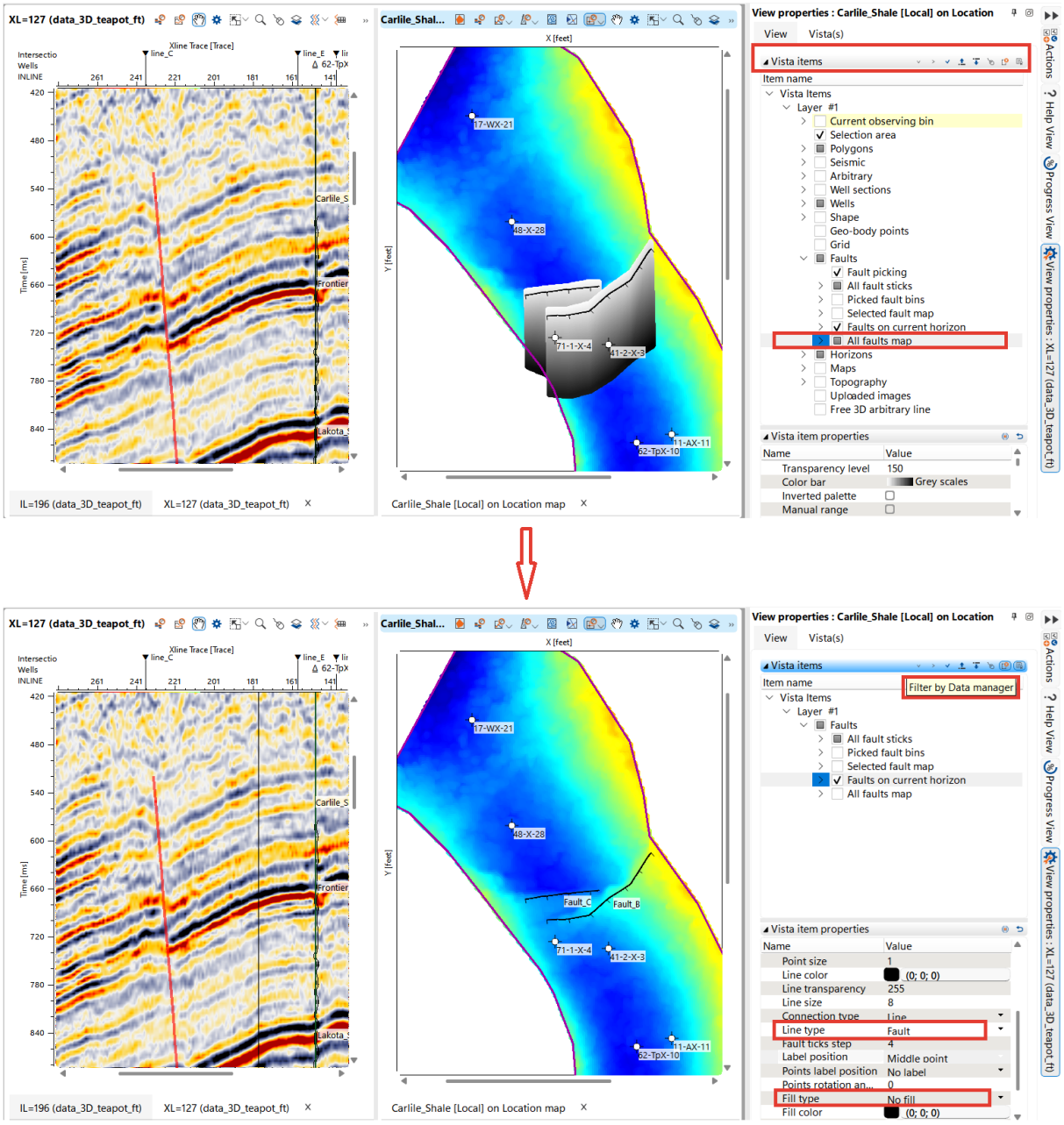


By default, in g-Space, all objects are displayed layer by layer in the map window, so the fault plane projection will be visible on the map. Users can filter and organize objects in the **Object tab** tools menu located on the top panel.

© 2026 Geomage 2003 LTD

To display faults on the horizon map as a line, select **No fill** option in the Object item properties and choose Fault line to create an image similar to the one shown below.

For more details on panel properties, refer to the Object(s) and Visual Settings sections.

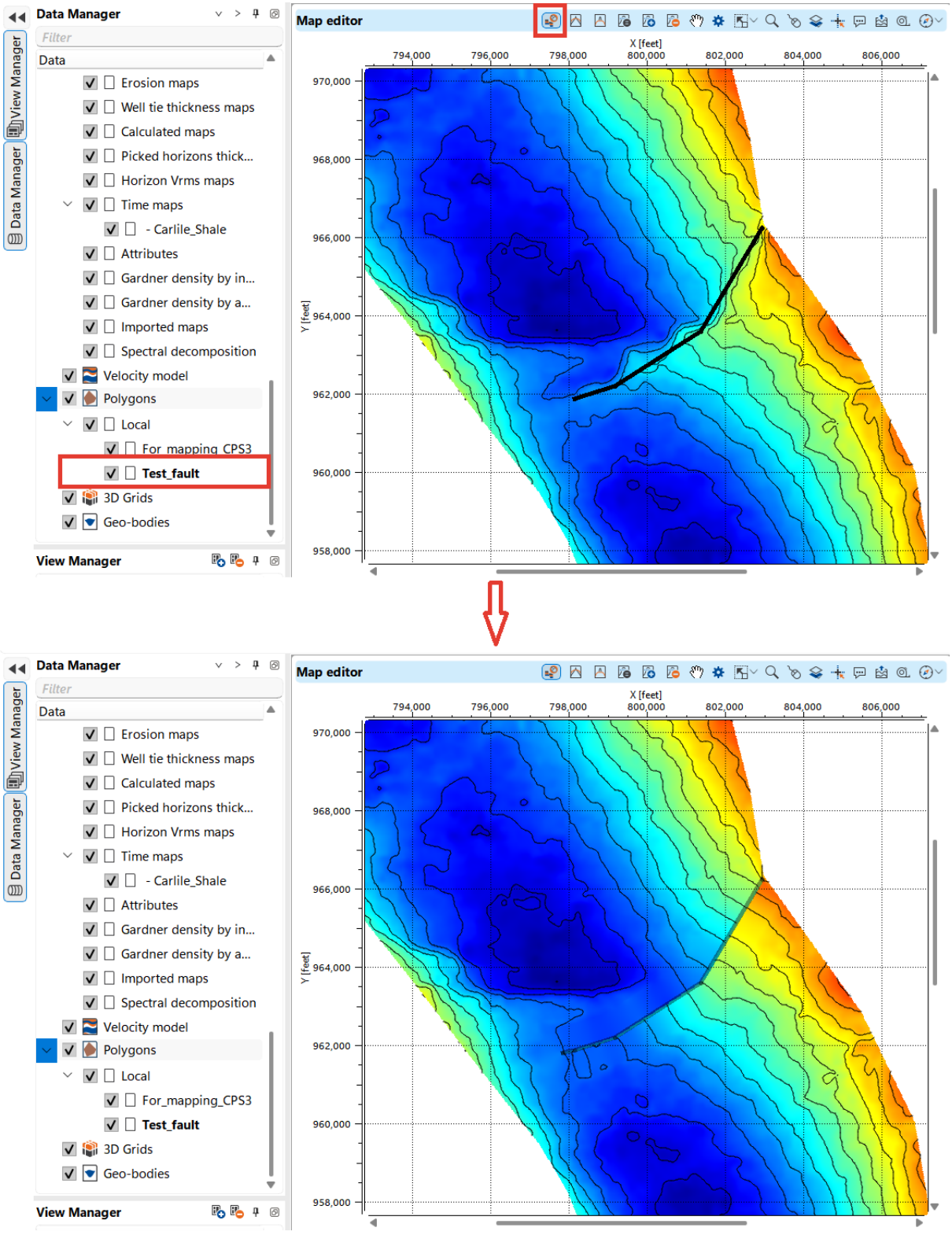


To rebuild a horizon map with fault constraints, open the map interpolation settings 100, set **Fault handling** to **Use selected faults** or **Use fault model**, and configure the corresponding faults. Then, in the **Data Manager**, right-click the horizon and select **Recreate horizon map**.

© 2026 Geomage 2003 LTD

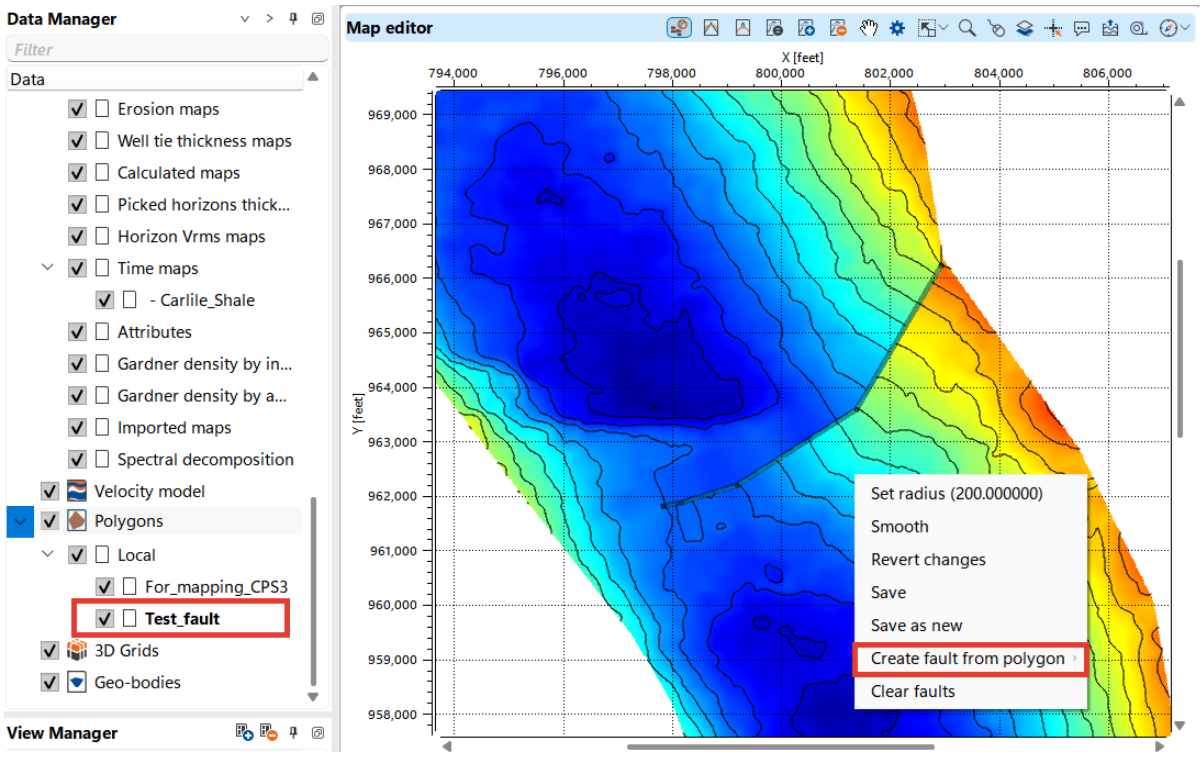
In the g-Space Map Editor 341, users can create a fault from a polygon. To do this, first create an empty polygon in the **Data Manager**. Then, draw the polygon on the map at the expected fault location, enabling the **Draw fault** function. Once you finish drawing by double-clicking, the polygon will resemble a fault, and the map will automatically update.

© 2026 Geomage 2003 LTD



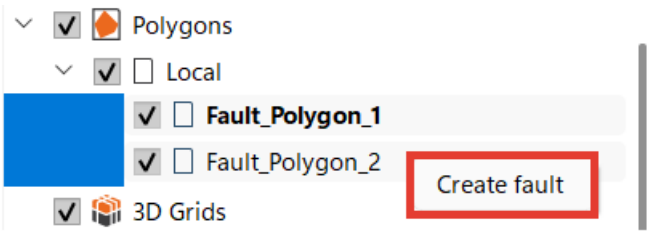
To save the polygon as a fault in the Data Manager, press **Alt + right-click** and select **Create fault from polygon**.

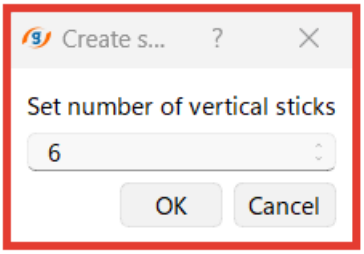
© 2026 Geomage 2003 LTD



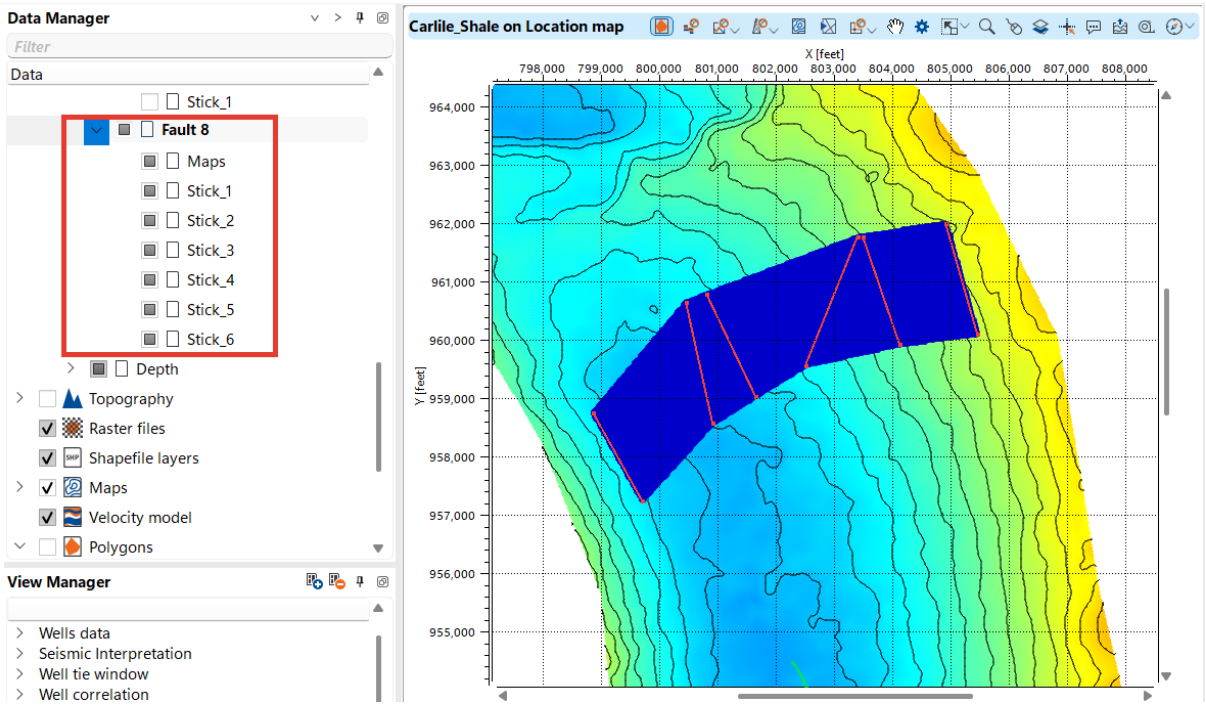
In the **Data Manager**, you can transfer data from polygons to faults by selecting the desired polygons, holding down the Shift key, and right-clicking. This will bring up the **Create fault** function in the menu. A pop-up will appear where you need to set the number of sticks for the break. We recommend setting the number of sticks to be no less than the number of polygon points.

© 2026 Geomage 2003 LTD



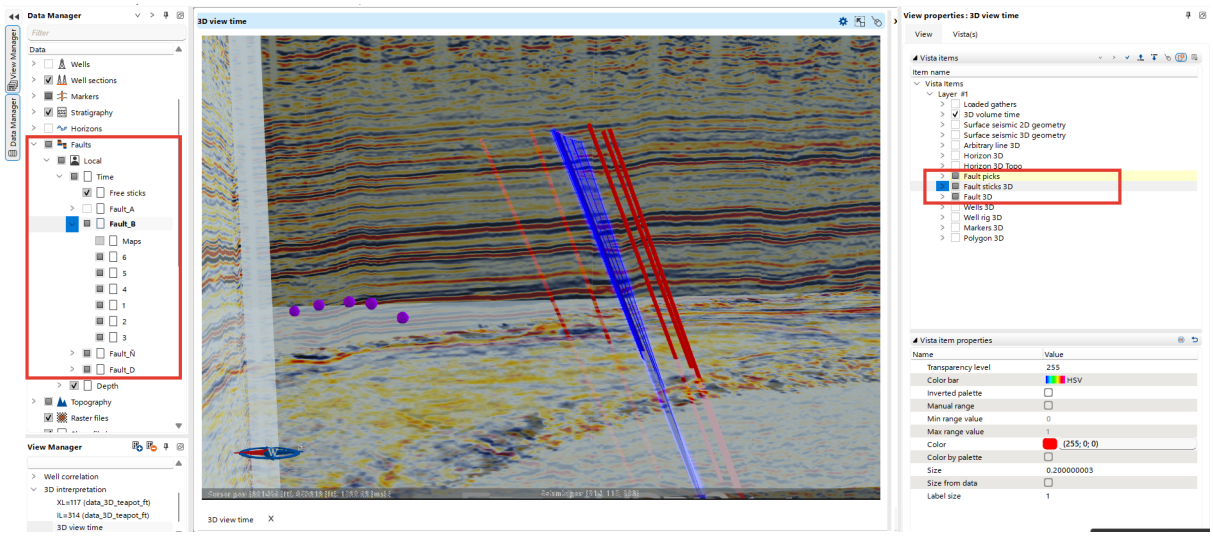


Name this fault and it will appear in the Data manager folder.

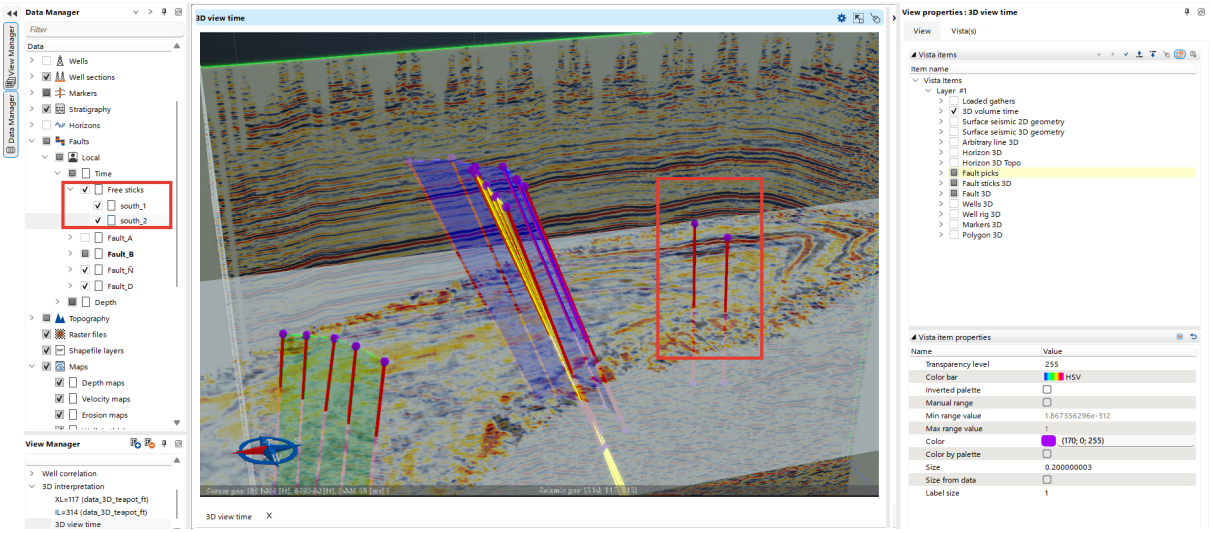


To work with faults in 3D view[244], you can set up visualization in the form of picks, sticks, fault maps and combine different types of visualization as shown in images below.

© 2026 Geomage 2003 LTD



The user can also work with free sticks and customize their visualization.



To export the created faults, use the **Faults Management** group on the Interpretation bar 96.

9.5.3.1 Fault prediction

The **Fault Prediction** tool in g-Space helps identify and analyze fault structures in seismic data. Access this tool by pressing **Alt + right-click** on any seismic section.

© 2026 Geomage 2003 LTD

Time Analysis Parameters

- Correlation time window: Defines the vertical analysis window for examining seismic trace patterns, measured in milliseconds. A larger window considers more data vertically.
- Correlation bin window: Determines how many adjacent bins (traces) to analyze horizontally. Larger values include more neighboring traces for pattern recognition.
- Maximum shift: Sets the maximum allowable time shift when correlating traces, measured in milliseconds. Limits the search range for matching patterns.

Visual Enhancement

- Time stretch: Controls the vertical exaggeration percentage of the display. Higher values enhance fault visibility by making them appear more vertical.

Smoothing Controls

- Sigma for local faults: Fine-tunes the smoothing for detailed fault features. Lower values maintain more local detail.
- Sigma for generalized faults: Controls regional fault trend smoothing. Higher values emphasize major fault systems.
- Sigma for noise reduction: Determines noise filtering intensity. Higher values produce cleaner images but may reduce subtle details.

Processing Control

- Autopicking step: Sets the increment between automatic fault picks. Lower values increase detail but require more processing time.

Saving Predictions

The right-click context menu on a time seismic section provides options to turn predictions into fault geometry:

- **Run fault prediction** - compute the fault prediction for the current section.
- **Create new stick from prediction** - build a single stick from the prediction at the clicked location.
- **Create new sticks from all predictions** - build sticks for every predicted fault on the section in one step and save them as free sticks (named `PredictedStick_1`, `PredictedStick_2`, and so on).

© 2026 Geomage 2003 LTD

9.5.3.2 AI Faults Wizard

AI Faults Wizard is a tool for automatic fault detection in 3D seismic data using neural networks. The wizard performs a complete processing cycle: from fault detection to generation of geometric objects ready for interpretation and analysis.

Purpose

The tool is designed for:
- Automating fault detection in seismic data
- Accelerating structural features interpretation
- Improving quality and detail of fault mapping
- Reducing subjectivity in manual interpretation

Features

- Automatic fault detection using trained neural networks
- Calculation of planarity (reflector continuity) and faultness (fault centerline) attributes
- Determination of fault orientation in space
- Generation of fault geometry (fault sticks) with automatic clustering into faults
- Preset parameters for quick start
- Filtering of the extracted faults by dip, azimuth and size before they are saved

Workflow

The wizard includes three sequential steps (tabs):

1. **AI Fault Attribute** — fault detection using neural network
2. **Attribute Processing** — calculation of planarity and fault characteristics (faultness and orientation)
3. **Sticks & Faults** — fault geometry construction

Requirements

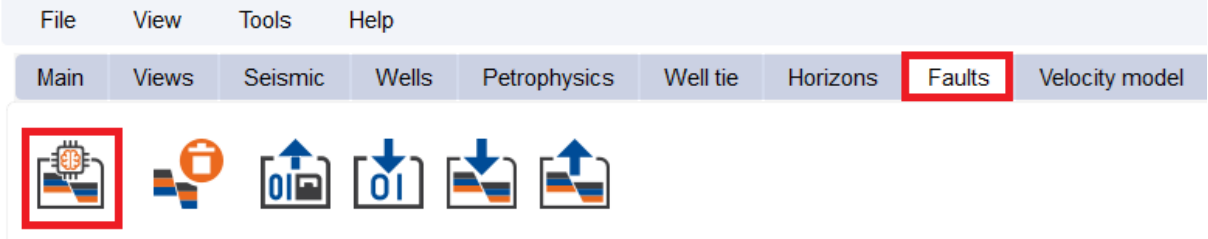
- 3D seismic cube in SEGY format loaded into the project
- Seismic data available in Data Manager
- Sufficient RAM for cube processing

Launching the Wizard

Launch methods:

© 2026 Geomage 2003 LTD

- Through **Wizards** panel (if activated)
- Through Ribbon 33 menu → **Faults** → **AI Faults Wizard**



The wizard opens as a dialog window with three tabs (**AI Fault Attribute**, **Attribute Processing**, **Sticks & Faults**). Each tab contains parameters for the corresponding processing stage.

Results:

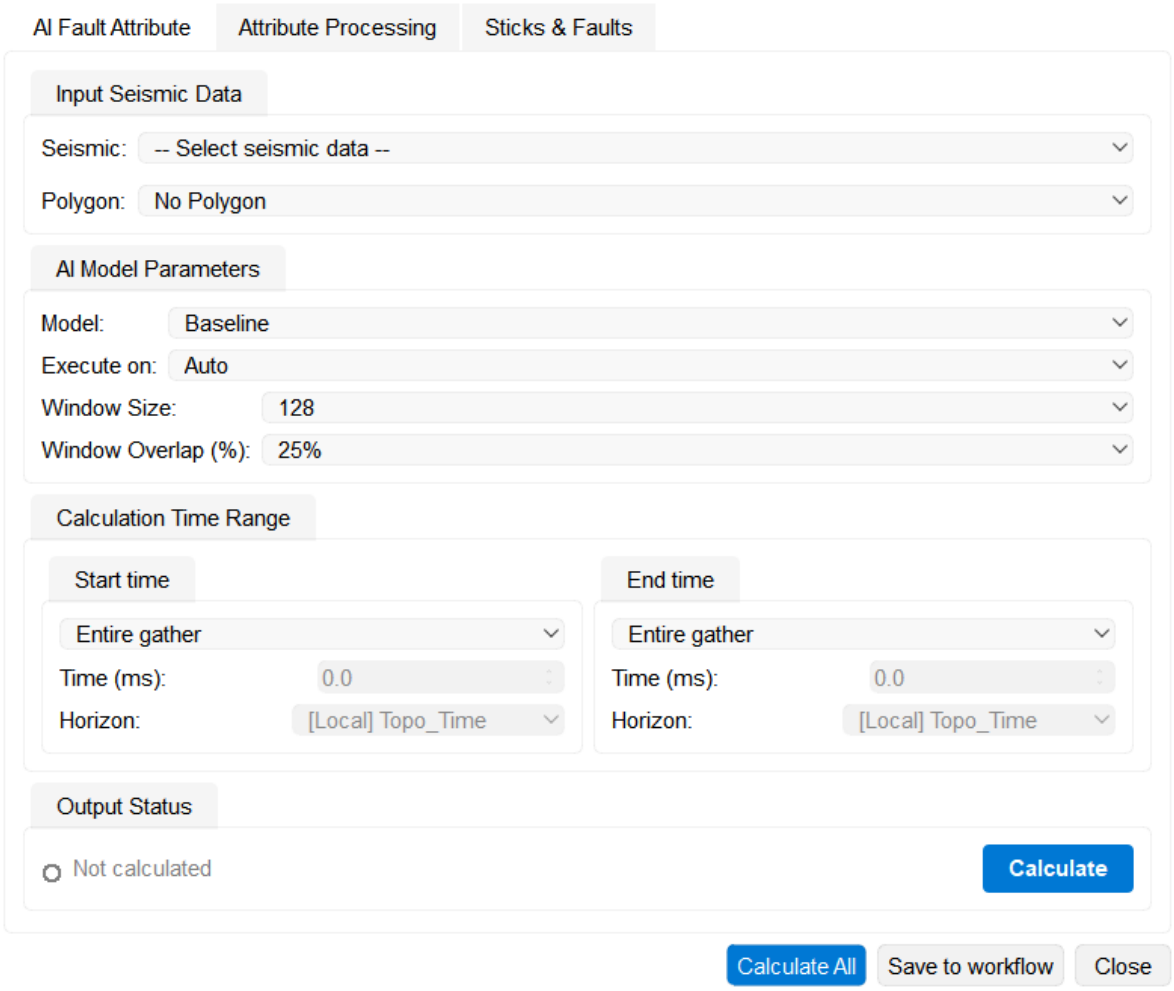
- All calculated attributes are automatically saved in Data Manager 124 in the **Seismic Data / Attributes** section
- Each attribute is assigned a suffix for identification: `_AIFaults`, `_Planarity`, `_Faultness`, `_Orientation`
- Visual indicators (green checkmark) show successful stage completion
- Results from one step automatically become available for selection in the next step

Step-by-Step Guide

AI Fault Attribute Tab: Fault Detection

Application of a trained neural network for automatic fault detection in seismic data. The neural network analyzes fracture patterns and creates a probability map of fault presence.

© 2026 Geomage 2003 LTD



Procedure:

1. Select input data

- In **Seismic:**, select an available 3D seismic cube. Two-dimensional data is not supported.
- In **Polygon:**, select **No Polygon** to process the full volume, or select an available polygon to restrict the attribute calculation, sticks and fault surfaces to that area.

2. Configure AI model parameters

Model: selects the trained network used to estimate fault probability:

- **Baseline** — the default model. It accepts every listed **Window Size**, so you can trade local detail and processing cost against broader structural context.
- **Transformer** — an alternative trained model with a fixed **Window Size** of **128**. Selecting it sets and disables the window-size control.

© 2026 Geomage 2003 LTD

Note: *Both models produce fault-probability attributes rather than interpreted fault objects. For a new dataset, compare their results on a representative polygon and use the model that best preserves the target fault patterns before processing the full volume.*

Execute on: controls the processing device:
- **Auto** — tries NVIDIA GPU acceleration when it is available and falls back to CPU if GPU startup fails (default)
- **CPU** — runs on the processor
- **NVIDIA GPU** — explicitly requests GPU acceleration; if it is unavailable, the warning offers actions such as running on CPU or opening gInstaller

Window Size: is the seismic cube window supplied to the model:
- **32** — smallest local context and lowest processing cost
- **64** — a larger local context with moderate processing cost
- **128** — default for Baseline and the fixed value for Transformer
- **192** — broader structural context with greater time and memory use
- **256** — broadest available context and greatest processing cost

Window Overlap (%): is the overlap between adjacent windows:
- **0%** — fastest, with the greatest risk of visible window boundaries
- **10%** — limited boundary smoothing with a small processing increase
- **25%** — default balance between smoothing and processing cost
- **50%** — greatest boundary smoothing and highest processing cost

Calculation Time Range

Use the separate **Start time** and **End time** selectors to restrict detection to a vertical interval. Each selector has these modes:
- **Entire gather** — no limit at that end of the trace
- **Time** — use the fixed value entered in **Time (ms)**
- **Horizon** — follow a time horizon selected from the project-dependent **Horizon:** dropdown

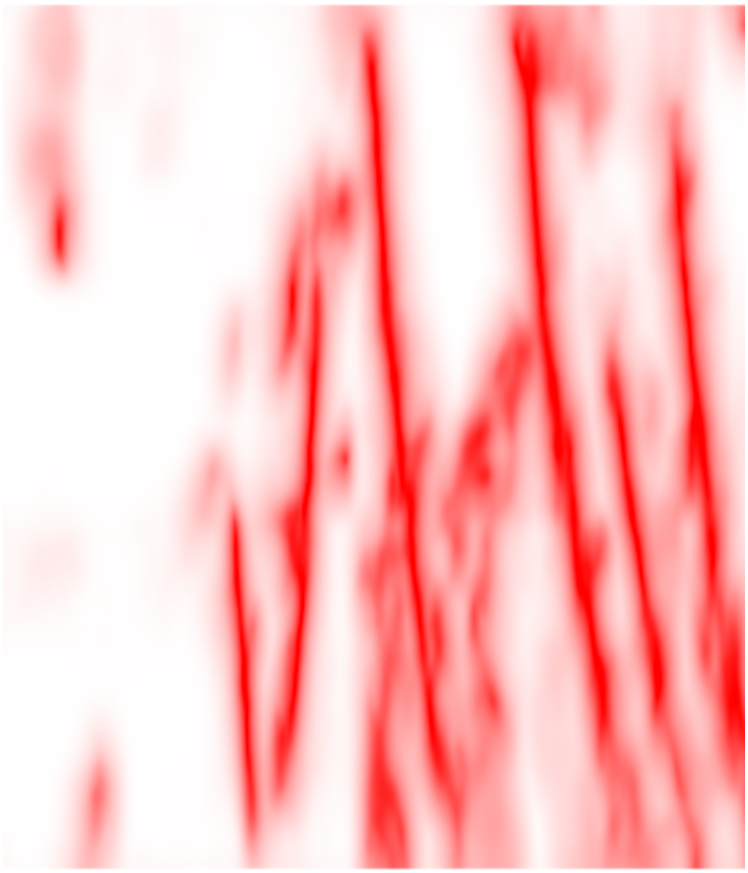
Leave both selectors at **Entire gather** to process the full trace length.

3. Run calculation
- In the **Output Status** group, press the **Calculate** button
- Processing may take from several minutes to several hours depending on cube size and parameters
- Result is automatically saved in Data Manager with suffix `_AIFaults`
- After completion, a green checkmark and the calculated attribute name appear in the **Output Status** group

Result:

© 2026 Geomage 2003 LTD

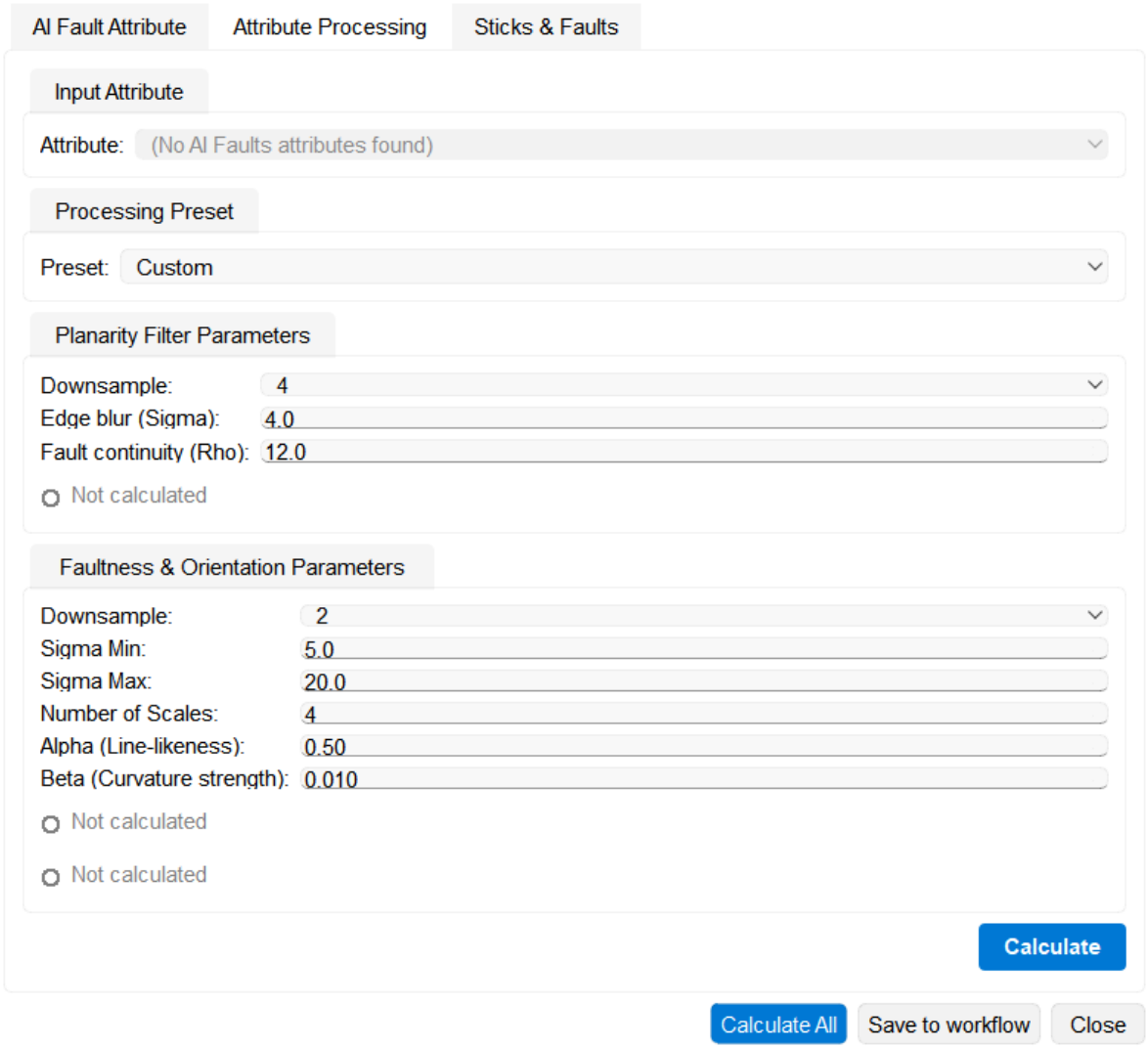
AI Faults attribute (SEGY format) — probability map of fault presence, where high values correspond to areas with high fault probability.



Attribute Processing Tab: Attribute Processing

Calculation of planarity and fault characteristics attributes. The tab contains two processing stages.

© 2026 Geomage 2003 LTD



Processing sequence

The tab first calculates **Planarity**, a measure of seismic-reflection continuity, and then uses that result to calculate **Faultness** and **Orientation**. Faultness highlights a fault centerline for stick construction; Orientation records its horizontal direction.

1. Select an attribute

- The **Attribute:** dropdown lists available AI Faults attributes.
- The result from the **AI Fault Attribute** tab is selected automatically; choose another compatible attribute when required.

2. Select a processing preset

© 2026 Geomage 2003 LTD

The single **Preset:** dropdown in **Processing Preset** controls both the planarity and faultness/orientation calculations:

- **Sharp** — limits smoothing for clear, well-defined faults
- **Balanced** — default starting point for general data
- **Smooth** — increases smoothing for noisy data or weak fault responses
- **Custom** — exposes all parameters in both groups for manual tuning

Note: *Start with Balanced. Compare Sharp and Smooth on a representative area when boundaries are respectively clear or noisy; use Custom after inspecting those results because its controls affect both continuity filtering and fault-centerline extraction.*

With **Custom**, configure:

- **Planarity Filter Parameters — Downsample: 1**, **2**, **4**, **6** or **8**; higher values reduce processing time and resolution
- **Edge blur (Sigma):** smoothing scale used when detecting edges
- **Fault continuity (Rho):** neighborhood scale used to evaluate structural continuity
- **Faultness & Orientation Parameters — Downsample: 1**, **2**, **4**, **6** or **8**
- **Sigma Min** and **Sigma Max:** lower and upper smoothing scales
- **Number of Scales:** number of scales included in the multi-scale calculation
- **Alpha (Line-likeness):** sensitivity to line-like structure
- **Beta (Curvature strength):** sensitivity to curvature strength

Note: *Use lower Downsample values to retain detail and higher values for quicker trials. Wider smoothing and continuity scales can stabilize noisy responses but can suppress local changes, so compare Custom settings on a small area before a full-volume run.*

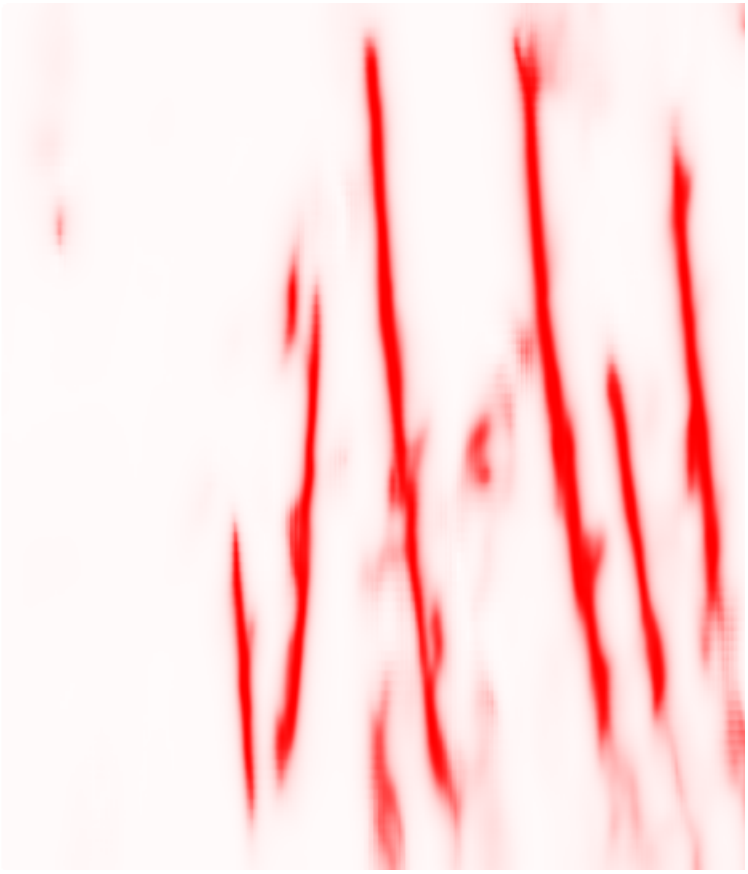
3. Run the calculation

- Press **Calculate**. The wizard calculates Planarity first, then Faultness and Orientation.
- The results are saved with `_Planarity`, `_Faultness` and `_Orientation` suffixes.
- Status indicators show the result of each calculation.

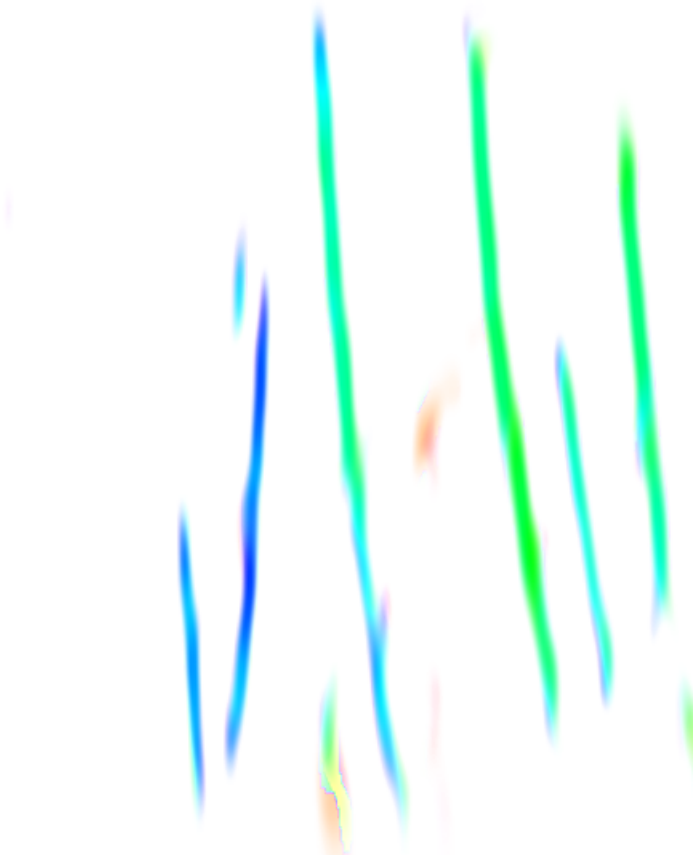
Results:

- Planarity attribute (SEGY) — planarity map
- Faultness attribute (SEGY) — fault centerline (ridge)
- Orientation attribute (SEGY) — fault strike azimuth in map view

© 2026 Geomage 2003 LTD



© 2026 Geomage 2003 LTD



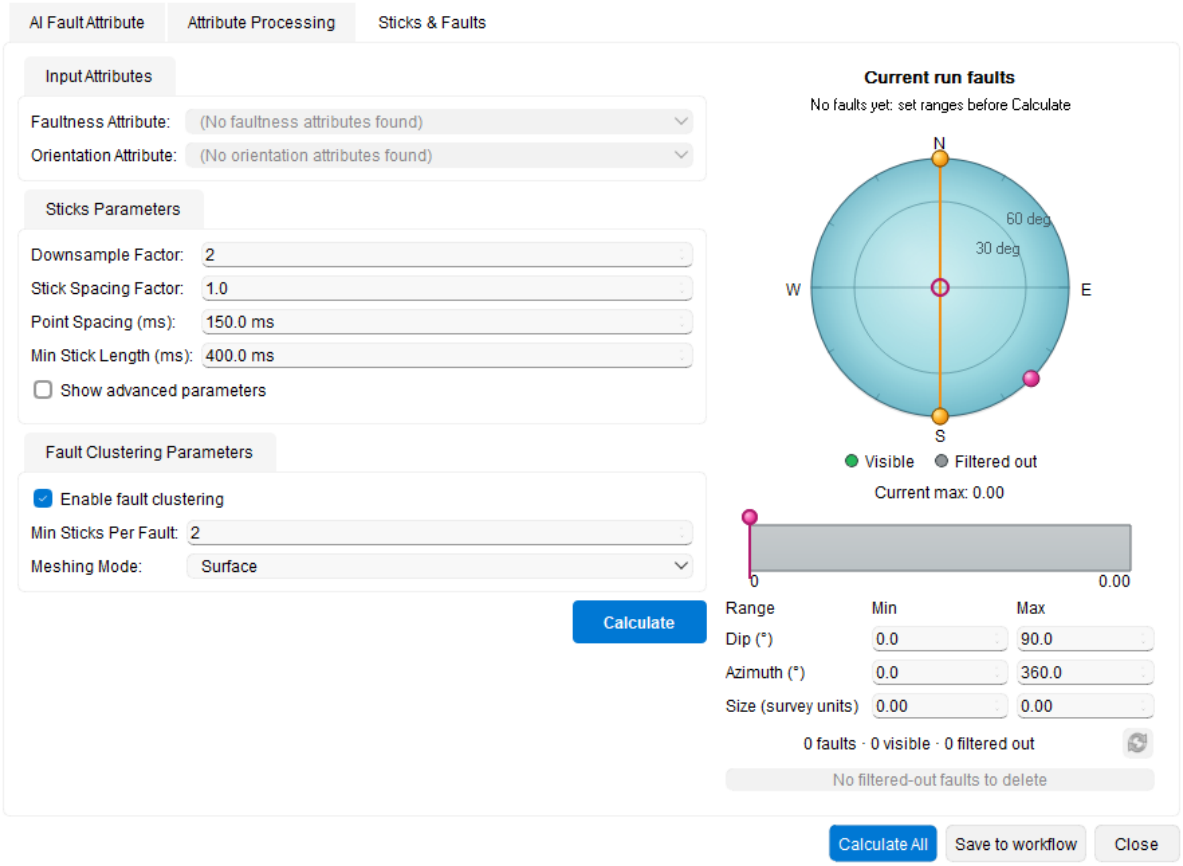
Sticks & Faults Tab: Fault Stick Generation

Construction of three-dimensional fault geometry (fault sticks) based on fault centerline (Faultness) and orientation (Orientation) attributes. Faultness highlights the fault axis (ridge), allowing automatic stick construction along the centerline.

Process consists of two stages:

1. **Stick tracing** — construction of vertical lines along fault central axes. Each stick extends while the fault remains pronounced (high Faultness) and the fault strike azimuth does not change more than the tolerance (Orientation Tolerance).
2. **Grouping into faults** — for each stick in each time slice, the algorithm searches for neighbors in two directions along the fault strike (along the high Faultness line). Sticks are combined into one fault only if they are mutual neighbors: if stick A sees stick B on the left, then stick B must see stick A on the right. This prevents erroneous merging at fault branches.

© 2026 Geomage 2003 LTD



Procedure:

1. Select attributes

- In the **Input Attributes** group, select the fault centerline attribute (Faultness) and orientation attribute (Orientation) from the Fault Attributes stage
- Both attributes are selected automatically if the **Attribute Processing** tab is completed

2. Configure parameters

In the **Sticks Parameters** group:

- **Downsample Factor:** reduces resolution for faster processing; **1** keeps full resolution
- **Stick Spacing Factor:** sets spacing as a multiple of fault thickness; **1.0** means the spacing equals the thickness
- **Point Spacing (ms):** sets the distance between points along a stick; **0** keeps all points
- **Min Stick Length (ms):** excludes shorter sticks
- **Show advanced parameters:** displays **Orientation Tolerance (deg)**
- **Orientation Tolerance (deg):** sets the maximum direction change allowed while extending a stick

In the **Fault Clustering Parameters** group:

© 2026 Geomage 2003 LTD

- **Enable fault clustering:** groups connected sticks into faults; clear it to retain free sticks
- **Min Sticks Per Fault:** sets the minimum group size; smaller groups remain orphan sticks
- **Meshing Mode — Surface:** reconstructs a continuous surface mesh
- **Meshing Mode — Sticks:** represents faults as individual stick lines

Note: *Higher downsampling or wider stick spacing is useful for faster trials but reduces geometric detail. Increase the minimum stick length to suppress short fragments; increase Orientation Tolerance only when valid curved faults are being broken into separate sticks.*

3. Set the fault filter

The right-hand column of the tab contains the **Dip Azimuth Filter**. It is headed **Current run faults** because it acts only on the faults produced by the current run of this wizard; faults that are already in the project stay untouched. A separate **Fault Filtering** window offers the same controls for all faults of the project.

Before the first calculation the compass is empty and shows the hint **No faults yet: set ranges before Calculate**. Ranges set at this point are applied during the extraction itself, so only the faults that pass them are created.

Set the limits in the **Range** grid, or directly on the plots:

- **Dip (°)** — minimum and maximum fault dip, from 0 to 90 degrees; on the compass, drag the pink handles radially.
- **Azimuth (°)** — the strike interval, from 0 to 360 degrees; on the compass, drag an orange boundary handle to resize the wedge and drag inside it to rotate. The interval may cross 0/360 degrees.
- **Size (survey units)** — minimum and maximum fault size. The strip above the grid shows the size distribution of the current faults with their **Current max** value; drag its pink handles to set the same limits.

The default ranges are fully open — dip 0 to 90, azimuth 0 to 360 and the whole size range — so a run in which the filter is left untouched saves every extracted fault. The reset button next to the fault counts (tooltip **Reset all filters**) restores these full ranges.

4. Run calculation

- Press the **Calculate** button (at the bottom of the tab)
- Results are automatically added to Data Manager (Faults / Time or Depth)
- Only the faults that pass the configured ranges are kept; the others are removed rather than hidden, and widening the ranges later does not bring them back
- The filter then reports how many faults were created and how many were removed by the filter

5. Review the result in the filter

© 2026 Geomage 2003 LTD

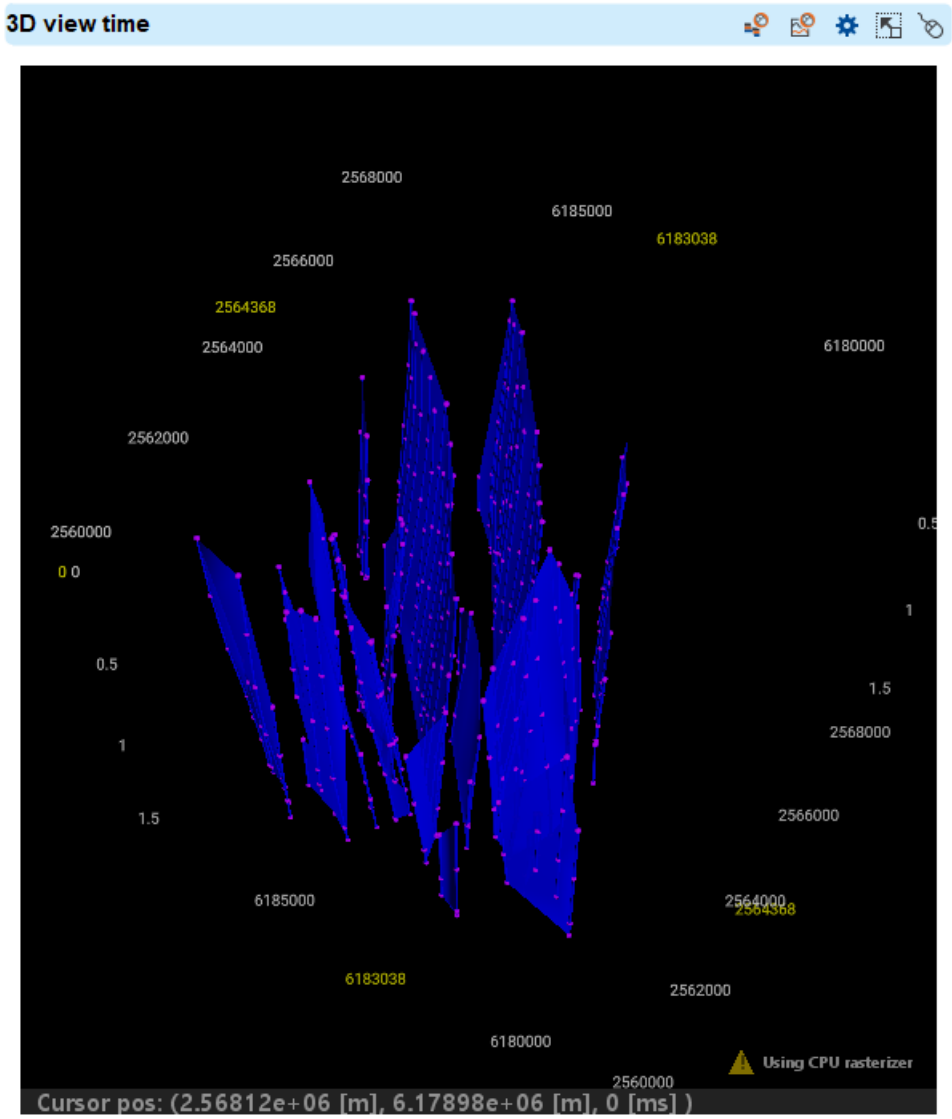
After the calculation the compass shows one marker per fault of this run; point at a marker to read its name, dip, azimuth and size. Faults that pass the current ranges are marked **Visible**, the others **Filtered out**, and the line under the grid counts the total, visible and filtered-out faults.

Changing the ranges now only previews the result. To remove the faults that no longer pass, press the delete button under the counts; it names the number of filtered-out faults, and reads **No filtered-out faults to delete** while every fault passes. If filtered-out faults are still pending when the wizard is closed, g-Space asks whether to delete them; keeping them leaves them in the project.

Result:
Fault Sticks — three-dimensional fault geometry represented by a set of vertical lines (sticks), grouped into individual faults. Available for:

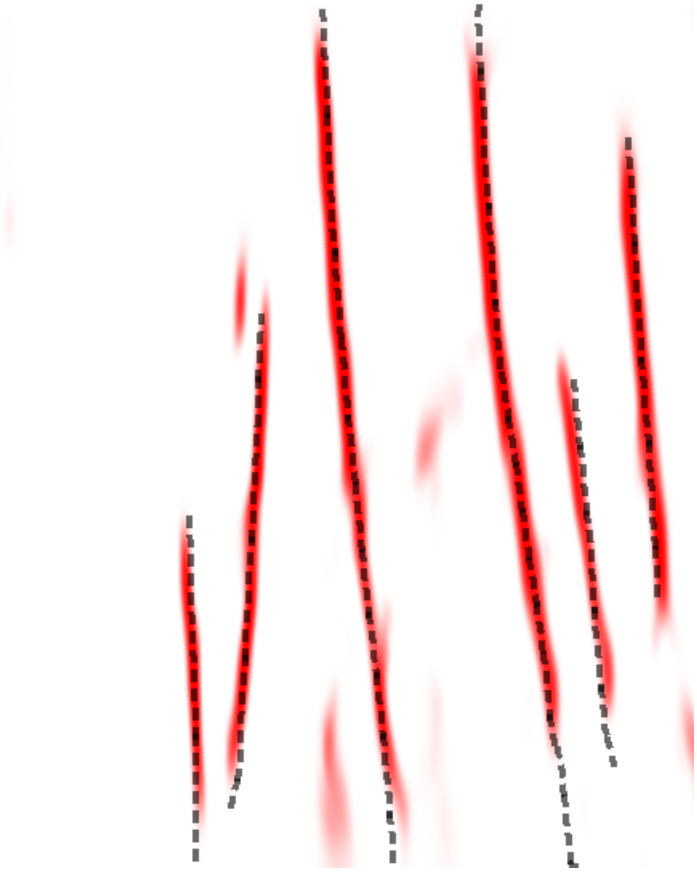
- Visualization in 3D View 244, Map View 235, Seismic Sections
- Conversion to Surface Mode 627 (fault surfaces)
- Export to other applications
- Further editing and interpretation

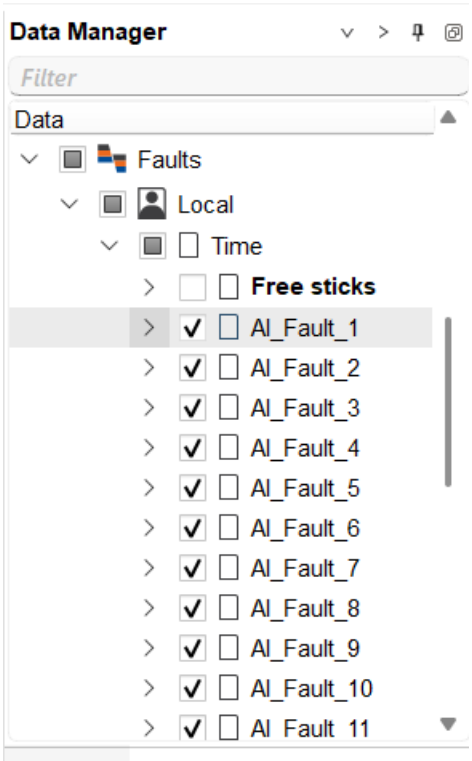
© 2026 Geomage 2003 LTD



3D view time ×

© 2026 Geomage 2003 LTD





© 2026 Geomage 2003 LTD

Calculate All

The **Calculate All** function performs all three steps sequentially in automatic mode — from fault detection to geometry generation. This is the most convenient method for complete processing.

Procedure:
1. Go to the **AI Fault Attribute** tab and select the seismic cube
2. Review and configure parameters on all three tabs (or leave defaults for quick start)
3. Press the **Calculate All** button (at the bottom of the wizard window, next to the **Close** button)
4. The wizard will automatically execute all stages in correct sequence

Advantages:
- Time saving: No need to manually switch between tabs
- Reliability: Automatic transfer of results between stages
- Convenience: Ideal for first run with default parameters

Save to workflow

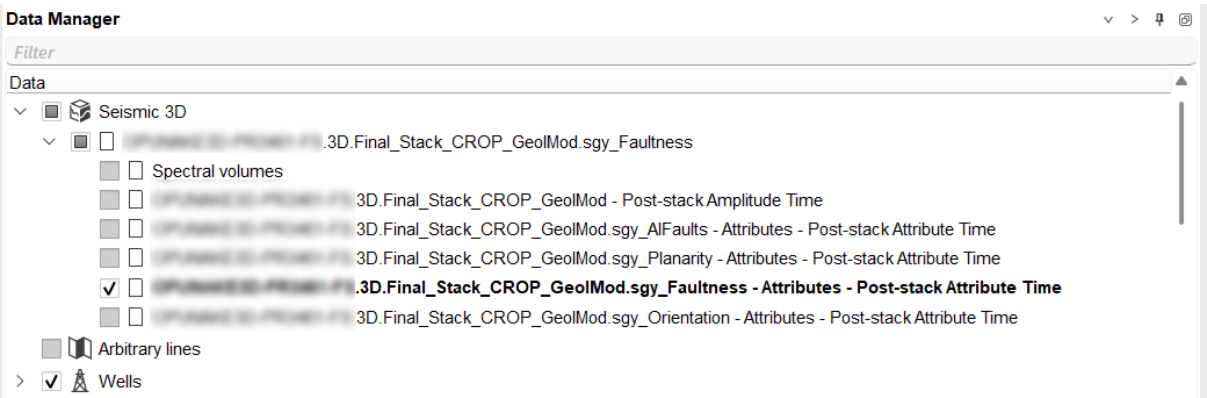
After configuring the inputs and parameters, click **Save to workflow** to store the current wizard settings as a workflow task. Saving does not run the calculation; the task can be opened or executed later from the workflow.

The dip, azimuth and size ranges of the **Dip Azimuth Filter** are stored with the task and restored when it is reopened. When the task runs from the workflow, the same ranges are applied and the faults that do not pass are not kept.

Visualization of Results

Viewing Attributes:
After executing **AI Fault Attribute** and **Attribute Processing**, attributes are available in Data Manager → **Seismic Data / Attributes**. Attributes can be visualized in 2D/3D Seismic Views.

© 2026 Geomage 2003 LTD



Viewing Fault Sticks:
Sticks & Faults results are available in Data Manager 124 → **Faults / Time (or Depth)**. Visualization options: 3D View 244, Map View 235, Seismic Sections. Mode switching: Free Sticks or Surface Mode 627. Display settings in Visual Settings 206 (color, thickness, transparency).

Recommendations

General Approach:
- Start with Preset=**Balanced** for first run
- Use **Calculate All** for automatic execution of all steps
- Follow iterative approach: quick assessment → result analysis → detailed processing
- Use polygons to limit processing area for parameter testing

Performance Optimization:
- For a quick Baseline assessment, use **Window Size 64** and **Window Overlap 10%**; Transformer always uses Window Size 128
- Use Downsample=2 or 4 on large cubes
- Increase Stick Spacing Factor to 2.0-3.0 for preview calculations

Quality Improvement:
- Check seismic data quality (structural smoothing can improve results)
- For clear faults: use Preset=**Sharp** with minimal smoothing
- For noisy data: use Preset=**Smooth** with increased regularization
- For curved faults: increase Orientation Tolerance to follow fault bends in horizontal plane

See Also

- Fault picking 627
- Seismic attributes 680
- Data Manager 124
- Import faults 441

© 2026 Geomage 2003 LTD

- Export faults 480

9.6 Velocity model workflow

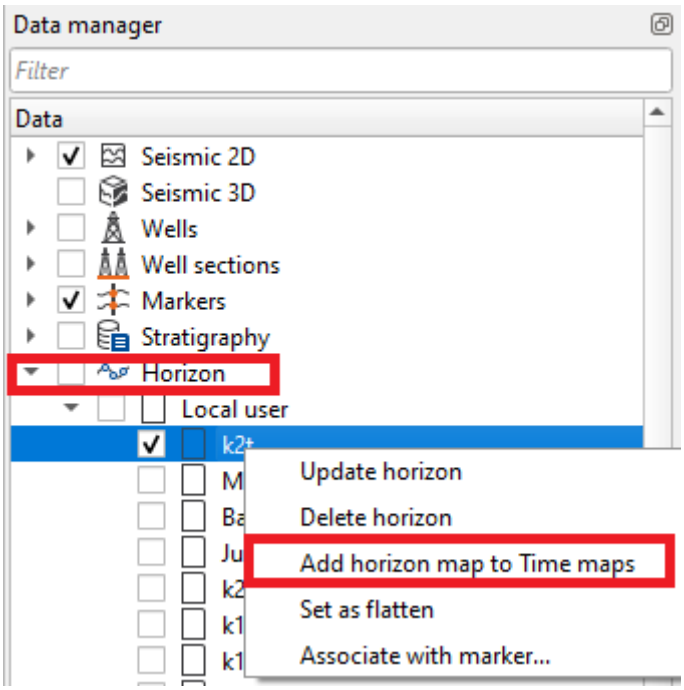
The **Velocity Modeling** in g-Space allows users to create and refine velocity models interactively, utilizing /// data.

In this guide, we will explore the various tools available under Velocity Modeling using the /// from the Demo project.

> *Subscribe to our channel and watch the video on how to build a velocity model in g-Space on YouTube*

Grid maps can be created in g-Space by converting horizons to time grids or running velocity model which stores all results in Maps folder in Data Manager 124

To convert horizon to time grid press Right Click Mouse on horizon to convert on Horizon icon in Data Manager and choose **Add horizon to Time Maps**



The **Data Manager** for velocity models in g-Space has a hierarchical structure to manage different velocity model types and their associated data. Here's a breakdown of the structure as shown in the image:

© 2026 Geomage 2003 LTD

1. **Velocity Model**: This is the top-level category where all the velocity models are organized.
2. Model name: This section holds data related to the specific velocity model.
   - **Velocity**:
     - **Interval Velocity**: Displays the constant velocity values used in the interval velocity calculation./ **Interval Velocity**: Shows interval velocity values for specific horizons (e.g., Carlile_Shale, Lakota_Sand, Tensleep_Sand).
     - **Average Velocity**: Displays the constant velocity values used in the average velocity calculation. /**Average Velocity**: Shows average velocity values for the same horizons.
   - **Skeleton**:
     - **Time**: Represents the time skeleton of the velocity model.
     - **Depth**: Provides depth information for the horizons.
   - **Depth**:
     - **By Interval Velocity**: Represents the depth calculated using interval velocity.
     - **By Average Velocity**: Represents the depth calculated using average velocity.
   - **Wells Mismatch**:
     - **Points**: Represents any mismatch between the well data and the velocity model.
   - **Map**:
     - **By Interval Velocity**: Shows interval velocity maps for horizons.
     - **By Average Velocity**: Shows average velocity maps for horizons.

Displaying model layers in depth sections

The depth surfaces of the active velocity model (the model set as default in the Data Manager) can be displayed on depth-domain seismic sections: **2D Line**, **3D Inline**, **3D Crossline** and **Arbitrary Line**. In the view they are listed as the **Depth average maps projection** object.

Use the check boxes in the Data Manager to control what is shown. Under **Skeleton** > **Depth**, select the check box of **- by average velocity** to draw all layers of the model on the active depth section, or select the check box of a single map to draw only that layer. Clear a check box to hide the corresponding surfaces.

Displaying model time maps in time sections

The time maps of the active velocity model can also be displayed on time-domain seismic sections: **2D line**, **3D inline**, **3D crossline** and **Arbitrary line**. In the view they are listed as the **Velocity model time maps projection** object, and each map is drawn as a line at the times it holds along the section.

Under **Skeleton** > **Time** in the Data Manager, select the check box of **Time** to draw all time maps of the model on the active time section, or select the check box of a single map to draw only that map. Clear a check box to hide the corresponding lines. Where a map has no values, its line is interrupted instead of being drawn across the gap.

© 2026 Geomage 2003 LTD

The surfaces of the active velocity model can also be shown in the time and the depth 3D view 244, where they are collected in the **Velocity model** folder of the **Object(s)** tab.

9.6.1 Setting up the velocity model

A **velocity model** describes how seismic velocity varies with position and depth across the survey. g-Space uses it to convert interpreted data between the time and depth domains, to build depth maps and cubes, and to support depth imaging and structural interpretation. This chapter describes how to launch the velocity modeling wizard and summarises the eight model-building methods it offers.

Launching the wizard

1. Open the Velocity model bar 99 in the ribbon.
2. Click **Create depth velocity**. The **Create velocity model** wizard opens.
3. On the first page, choose one of the velocity model types listed under **Velocity models**. A preview image illustrates the selected method.
4. Fill in the parameters and any layer or tie tables that appear for the chosen method, then click **Run** to build the model, or **Save to workflow** to keep the settings as a workflow task.

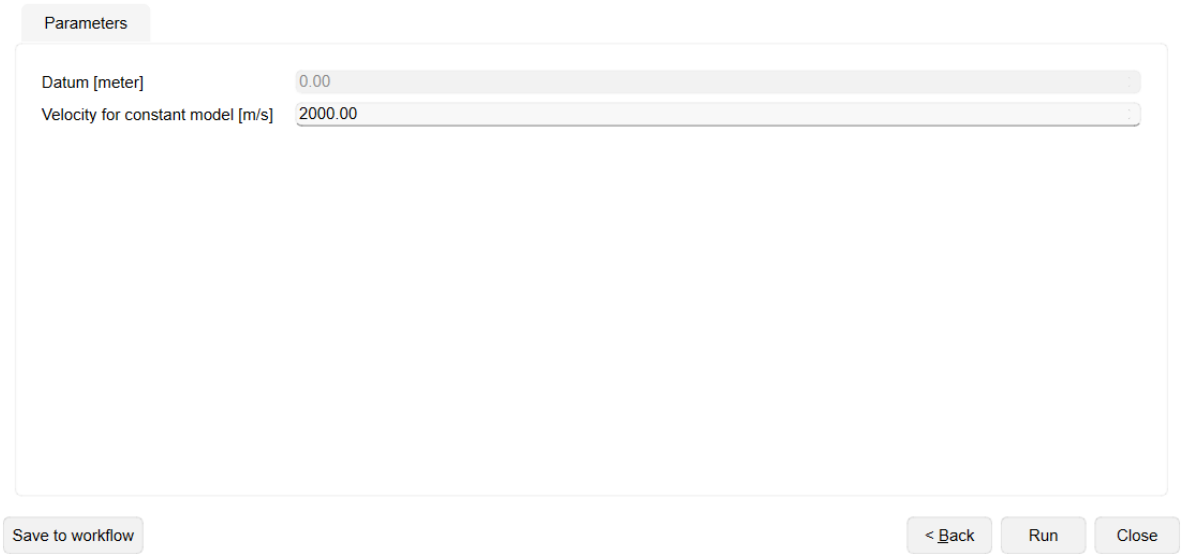
The new model is added to the depth velocity model set and becomes available for time-to-depth conversion and QC. Existing models can be brought in instead with Import velocity model 678.

Running the wizard or saving it to a workflow

The buttons along the bottom of the wizard decide what happens to the settings you entered:

- **Back** and **Next** — move between the pages of the wizard.
- **Run** — build the model now. You are asked for a name for the new velocity model, and if a model with that name already exists you are asked whether to overwrite it.
- **Save to workflow** — keep the current settings as a **Create velocity model** task instead of building the model now. The wizard closes, you are asked for the name of the velocity model the task will produce, and the task is added to the current workflow under that name.
- **Close** — close the wizard without building a model.

© 2026 Geomage 2003 LTD



A saved task stores the complete wizard setup — the selected velocity model type and every parameter entered for it. Reopen the task from the Workflows 535 wizard to review or change it: the **Create velocity model** wizard opens again with the saved values and keeps the same buttons. **Save to workflow** then writes the changes back into the same task instead of adding a second one, while **Run** builds the model straight away from the values on screen.

When the workflow itself is executed, the task builds the velocity model under its stored name without prompting. A velocity model that already carries that name is replaced without a confirmation question, so give the task a distinct name when earlier results must be kept.

Velocity model methods

The wizard offers the following methods, ordered from the simplest to the most data-intensive:

- Constant 662 — applies a single interval velocity to the whole output cube.
- Constant layered 663 — assigns a constant velocity to each layer, where layers are bounded by horizons, time maps, constant times or topography.
- Markers + horizons 664 — derives interval velocities from well marker-to-horizon ties and interpolates them between wells.
- Grid velocity by markers + horizons 666 — uses the same marker-to-horizon ties but writes the result on a bin grid.
- Checkshot extrapolation 667 — builds a layered cube from well checkshot surveys and extrapolates it across the grid.

© 2026 Geomage 2003 LTD

- Checkshot + markers 669 — combines checkshot depth-time calibration with well-marker depth control.
- Checkshot + markers + horizons 670 — combines checkshots, well markers and time horizons in a single well-tied model.
- Velocity volume 672 — builds an interval velocity cube from a velocity volume that is already in the project, in the time or in the depth domain.

See Also

- Velocity model bar 99
- Velocity model QC 675
- Import velocity model 678
- Workflows 535

9.6.1.1 Constant

The **Constant** method builds the simplest possible velocity model: a single interval velocity applied to the full output cube. It is used to produce an initial time-to-depth relationship when no detailed velocity information is available, or as a sanity reference to compare against more elaborate models.

Required inputs

- **Datum** — the elevation reference for the model. Taken from the project datum; read-only for this method.

Parameters

- **Velocity** — the constant interval velocity (m/s) applied to every sample of the cube. Default: 2000 m/s.

Workflow

1. Open Setting up the velocity model 660 and select **Constant** as the creation method.
2. Set the Datum if the default project value needs to be overridden.
3. Enter the constant velocity value.
4. Run the model. The wizard builds interval velocity, average velocity, time, and depth maps filled with the specified value.

© 2026 Geomage 2003 LTD

Output

A uniform velocity model consisting of interval velocity, average velocity, time and depth map stacks. All layers carry the single velocity value supplied in the wizard.

When to use

- Feasibility studies where only an order-of-magnitude depth conversion is needed.
- A baseline against which more elaborate velocity methods can be compared.
- Synthetic or training projects with no well control.

Avoid this method for production depth conversion — any real depth trend in the section will be flattened.

See Also

- Setting up the velocity model 660
- Constant layered 663
- Velocity model QC 675

9.6.1.2 Constant layered

The **Constant layered** method builds a velocity cube composed of piecewise-constant layers. Each layer is bounded by a user-selected surface (horizon, time map, constant time, or topography) and carries its own interval velocity. Useful when the geology is naturally divided into zones with distinct, near-uniform velocities.

Required inputs

- **Datum** — elevation reference for the model.
- **Layer table** — one row per layer. Each row specifies the **bottom of layer** (a horizon, time map, constant time, or topography) and the **velocity** to apply inside that layer.
- **Polygon** (optional) — restricts the model to the area enclosed by the polygon.

Parameters

All parameters of this method are defined through the layer table. No additional smoothing or interpolation options are exposed — the velocity stays strictly constant within each layer.

© 2026 Geomage 2003 LTD

Workflow

1. Open Setting up the velocity model 660 and select **Constant layered**.
2. Set the Datum and, if needed, select a polygon to bound the model area.
3. Populate the layer table. Use **Add Row** to add a layer, choose its bottom surface from the project, and enter the interval velocity.
4. Repeat for every velocity zone from top to bottom of the section.
5. Run the model. The wizard creates depth, interval velocity, average velocity, and time maps per layer.

Output

A layered velocity cube with interval velocity, average velocity, time and depth maps. Layer boundaries follow the selected surfaces exactly; velocity is discontinuous across boundaries and constant within each layer.

When to use

- Geologies with distinct velocity provinces separated by mapped horizons (for example salt/sediment, basalt/sediment).
- Quick time-to-depth conversion in layered basins when well control is sparse but structure is known.
- Initial model that will be refined later with checkshot- or marker-based methods.

Not suitable when velocity varies strongly inside a layer — use Checkshot extrapolation 667 or a marker-based method instead.

See Also

- Setting up the velocity model 660
- Constant 662
- Checkshot extrapolation 667
- Velocity model QC 675

9.6.1.3 Markers + horizons

The **Markers + horizons** method derives interval velocities from well-marker depths tied to time horizons, then interpolates spatially across the survey. Each marker-horizon tie defines a

© 2026 Geomage 2003 LTD

depth-time pair at a well location; the method converts these ties into layer velocities and fills the grid between wells via kriging or triangulation.

Required inputs

- **Datum** — elevation reference.
- **Marker-to-horizon tie table** — each row links one **Time horizon** to a **Marker / Group** picked on one or more wells.

Parameters

- **Max Depth** — lower bound of the output cube (default 4000 m).
- **Depth Step** — vertical sampling of the output (default 10 m).
- **Bottom Velocity** — fallback velocity applied below the deepest tie (default 5000 m/s).

Workflow

1. Open Setting up the velocity model 660 and select **Markers + horizons**.
2. Set Datum, Max Depth, Depth Step and Bottom Velocity.
3. Fill the tie table: for each time horizon pick the marker (or marker group) whose depth identifies it at the wells.
4. Optionally choose a trend per row to guide lateral interpolation (Vrms, layer thickness, or a data map).
5. Run the model. Velocities are computed per tie, then kriged/triangulated across the seismic trace grid.

Output

Interval velocity and average velocity cubes sampled on the seismic trace grid, plus depth and time maps at each time horizon. Velocity tracks the structure defined by the horizons while honouring the depth picks at wells.

When to use

- Projects with consistent marker picks across several wells and reasonably dense time horizons.
- Regional or exploration studies where checkshots are scarce but markers are plentiful.

© 2026 Geomage 2003 LTD

- Models that must honour geological structure between wells.

If the output must align to a regular bin grid or seismic cube geometry, use Grid velocity by markers + horizons 666 instead.

See Also

- Setting up the velocity model 660
- Grid velocity by markers + horizons 666
- Checkshot + markers + horizons 670
- Velocity model QC 675

9.6.1.4 Grid velocity by markers + horizons

The **Grid velocity by markers + horizons** method uses the same marker-horizon ties and the same wizard inputs as the Markers + horizons 664 method, but writes the result on a bin grid rather than on the native trace grid of the horizons. Use it when downstream tools need a regular, bin-aligned velocity volume.

Required inputs

- **Datum** — elevation reference.
- **Marker-to-horizon tie table** — time horizon + marker/group pairs; same structure as in Markers + horizons.

Parameters

The wizard page for this method shows the same parameters as Markers + horizons:

- **Max Depth** — lower bound of the cube (default 4000 m).
- **Depth Step** — vertical sampling (default 10 m).
- **Bottom Velocity** — extrapolation velocity below the deepest tie (default 5000 m/s).

The bin grid on which the cube is sampled is chosen when the model is run, not on the wizard page (see the workflow below).

Workflow

1. Select **Grid velocity by markers + horizons** in the Setting up the velocity model 660 wizard.
2. Set Datum, Max Depth, Depth Step and Bottom Velocity.

© 2026 Geomage 2003 LTD

3. Populate the marker-to-horizon tie table.
4. Run the model. g-Space asks whether to build the velocity cube on an external bin grid: choose an existing bin-grid item file, or let g-Space create a bin grid, and the velocities are then sampled onto that grid.

Output

Interval and average velocity cubes sampled on the chosen bin grid, plus depth and time maps. The lateral node spacing follows the bin grid selected when the model was run.

When to use

- When downstream tools (3D-grid creation, reservoir modelling) require a regular, bin-aligned velocity cube.
- When the velocity cube must align with a specific bin grid rather than the native horizon trace grid.

For models tied to the native horizon trace grid, use Markers + horizons 664 instead.

See Also

- Setting up the velocity model 660
- Markers + horizons 664
- Checkshot + markers + horizons 670
- 3D grid creating 765

9.6.1.5 Checkshot extrapolation

The **Checkshot extrapolation** method turns well checkshot surveys into a full 3D interval velocity cube. The user divides the section into layers bounded by time horizons and selects a velocity source for each layer (either a constant value or a checkshot-derived curve). The wizard interpolates velocities between wells and extrapolates them into the rest of the grid, optionally filtering the result to remove high-frequency artefacts.

Required inputs

- **Checkshot data** — loaded on one or more wells.
- **Layer table** — one row per layer, specifying:
- **Top / Bottom of layer** — time horizons, time maps, constant time or topography that bound the layer.

© 2026 Geomage 2003 LTD

- **Layering type** — **Proportional**, **Parallel to Top** or **Parallel to Bottom**; controls how the layer is subdivided internally.
- **Velocity definition** — **Constant** (user-entered value) or **From checkshot** (derived per well from the checkshot curve).

Parameters

- **Datum** — elevation reference.
- **Max Depth**, **Depth Step**, **Bottom Velocity** — output cube extent.
- **Frequency Filter** — high-cut filter applied to the checkshot velocity curve (default 25 Hz); smaller values give more aggressive smoothing.
- **Reference Geometry** — Maps intersection / Polygon / Seismic 3D, with the corresponding XY Step or IL/XL Each selectors.
- **Use Smooth** — enables post-interpolation smoothing, with **Smooth traces radius** and **Smooth samples radius** controls.

Workflow

1. Select **Checkshot extrapolation** in the velocity model wizard.
2. Configure Datum, Max Depth, Depth Step, Bottom Velocity, Frequency Filter and Reference Geometry.
3. Build the layer table by selecting top and bottom bounds, layering type and velocity source for each layer.
4. Enable post-smoothing if the output appears noisy.
5. Run the model. Per-well velocities are extracted from the checkshot, spatially interpolated, and written to the output cube.

Output

Interval and average velocity cubes, depth and time maps per layer, plus per-well mismatch logs between the model and the input checkshot (useful for QC).

When to use

- Projects where checkshot coverage is the primary velocity constraint.
- Models where geological layering is well understood and can be captured by horizons.
- Areas needing smooth, extrapolated velocities outside the well control.

© 2026 Geomage 2003 LTD

When markers or horizons give additional constraints, combine methods via Checkshot + markers 669 or Checkshot + markers + horizons 670.

See Also

- Setting up the velocity model 660
- Checkshot + markers 669
- Checkshot + markers + horizons 670
- Velocity model QC 675

9.6.1.6 Checkshot + markers

The **Checkshot + markers** method blends absolute depth-time calibration from checkshot surveys with intermediate depth picks supplied by well markers. Markers anchor velocity where checkshot sampling is sparse; checkshot calibrates absolute times. A kriging-based interpolator combines both sets of constraints into a single interval velocity cube.

Required inputs

- **Checkshot data** — one or more wells with depth-time tables.
- **Markers** — picks available on the same or on neighbouring wells.
- **Constant Velocity** — fallback value applied where neither checkshot nor markers constrain the grid.

Parameters

- **Datum**, **Max Depth**, **Depth Step** — output cube extent and sampling.
- **Topography type** — Internal or External topography.
- **Min / Max Velocity** — hard bounds clamping the output cube to physical values.
- **Mapping parameters:** control the well-tie interpolation.
- **Max marker deviation from well** — search radius used to associate markers with wells (default 50 m).
- **Well Tie Interpolation type** — Voronoi / ABOS / Triangulation / Kriging (default Kriging).
- **Trend smooth** — spatial smoothing radius applied to the trend (default 1000 m).
- **Kriging Covariance** — Spherical / Gaussian / Exponential, plus **Kriging Range** (correlation distance).
- **Marker type interpolation** — Linear or Regression; the regression mode adds a polynomial trend with user-supplied **Regression coefficient**.
- **Extrapolate Checkshot** and **Extrapolate Checkshot to 0** — extend checkshot-derived velocity below the last layer and up to the surface.

© 2026 Geomage 2003 LTD

- **Max distance to layer** — wells further than this distance from the current layer are excluded.

Workflow

1. Select **Checkshot + markers** in the wizard.
2. Configure Datum, Max Depth, Depth Step, Constant Velocity, Topography, Min/Max Velocity.
3. Open the Mapping parameters section and choose an interpolation type and kriging settings.
4. Enable checkshot extrapolation options if the model must extend beyond the checkshot sampling.
5. Run the model. Interval velocities are computed from checkshot, corrected by markers, and interpolated into the grid.

Output

Interval and average velocity cubes, depth/time maps, per-well tie-quality metrics and mismatch logs at every marker used in the calculation.

When to use

- Wells carrying both checkshot and marker data.
- Areas where markers give lateral control that pure checkshot extrapolation cannot reach.
- When depth-time calibration and structural constraint need to be honoured in the same model.

If time horizons are available as additional constraints, prefer Checkshot + markers + horizons 670 — it uses all three data types together.

See Also

- Setting up the velocity model 660
- Checkshot extrapolation 667
- Checkshot + markers + horizons 670
- Velocity model QC 675

9.6.1.7 Checkshot + markers + horizons

The **Checkshot + markers + horizons** method is the most comprehensive velocity-building tool in g-Space. It combines three data types: checkshot surveys for absolute depth-time

© 2026 Geomage 2003 LTD

calibration, well markers for lateral velocity variations, and time horizons for structural framework. All three are fed into a single kriging-based solver that produces a layered, well-tied interval velocity cube.

Required inputs

- **Checkshot data** — on one or more wells.
- **Markers** — picks on wells, typically one per stratigraphic unit.
- **Time horizons** — bounding surfaces for the layers.
- **Marker-to-horizon tie table** — each row links a time horizon to a marker or marker group. A **Trend** column controls how velocity is guided laterally: None, Vrms, Layer width, or Data map, with optional **inverse trend** and **Use markers inside group only** flags.

Parameters

- **Datum**, **Max Depth**, **Depth Step**, **Bottom Velocity**, **Constant Velocity** — output extent and fallback velocities.
- **Topography type**, **Min / Max Velocity** — same role as in Checkshot + markers.
- **Min thickness for velocity calculation** — skip layers thinner than this threshold.
- **Compute mismatch for all wells** — produce tie-quality logs for every well, not only the ones used in calibration.
- **Mapping parameters:** same interpolation controls as in Checkshot + markers (Max marker deviation, Well Tie Interpolation type, Trend smooth, Kriging Covariance and Range, Marker type interpolation, Regression coefficient, Extrapolate Checkshot, Max distance to layer, Use wells and horizons inside polygon).
- **Calculate velocity by checkshots** — if enabled, velocity is derived directly from the checkshot curve; otherwise the marker-based formulation is used.
- **Add marker horizons** — create interpreted horizons at marker depths and include them in the output.
- **Use marker projection if marker not exists** — project nearby marker picks when the marker is missing at a well.
- **VrmsStep** — sampling step for Vrms-based trend constraints.

Workflow

1. Select **Checkshot + markers + horizons** in the wizard.
2. Configure cube extent (Datum, Max Depth, Depth Step, Bottom / Constant Velocity), topography and velocity bounds.

© 2026 Geomage 2003 LTD

3. Populate the marker-to-horizon tie table. For each row choose a horizon, a marker and optionally a trend.
4. Configure the Mapping parameters (interpolation, kriging, marker smoothing).
5. Decide whether velocity should be driven by checkshots or markers via Calculate velocity by checkshots.
6. Enable QC and projection options as required and run the model.

Output

Layered interval velocity and average velocity cubes tied to the time horizons, depth and time maps, optional marker horizons, per-well time-depth mismatch logs and pine charts for quality control.

When to use

- Mature fields where all three data types are available.
- Production velocity models used for time-to-depth conversion and reserves estimation.
- Projects requiring rigorous QC via per-well mismatch logs and marker-horizon consistency.

If any of the three data types is missing, choose the method that uses only the available combination (for example Checkshot + markers 669 or Markers + horizons 664).

See Also

- Setting up the velocity model 660
- Checkshot + markers 669
- Markers + horizons 664
- Velocity model QC 675

9.6.1.8 From Vrms

The **Velocity volume** method builds a depth interval velocity model from a velocity volume that already exists in the project. The volume can be a 2D line or a 3D cube, in the time or in the depth domain, and it can hold RMS, interval or average velocities: you declare which type it contains and g-Space converts it accordingly. Use it when seismic velocities are the main constraint and well control is sparse.

The method was previously called **From Vrms** and merged every RMS velocity data set found in the project; it now converts the single volume you select.

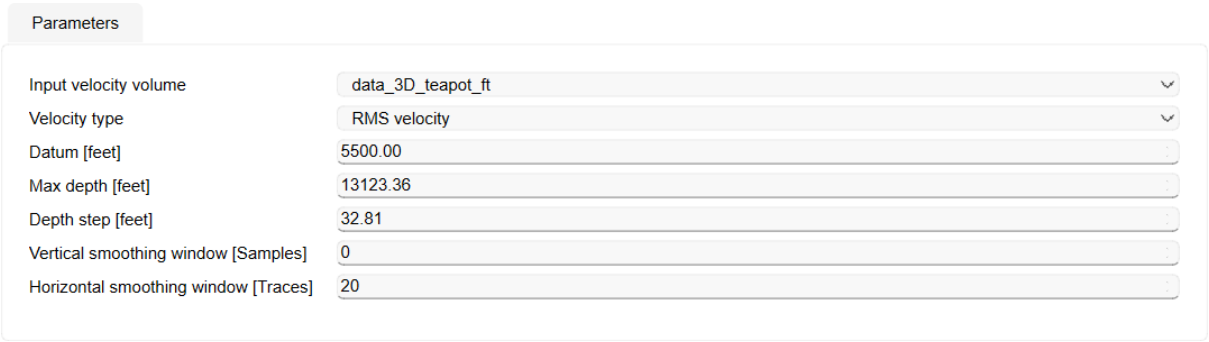
© 2026 Geomage 2003 LTD

Required inputs

- **Velocity volume** — one 2D line or 3D cube whose data type is Velocity, in the time or in the depth domain. Only the volume you select is used.
- **Time horizons** (optional) — smoothing targets for the horizontal smoothing of a time-domain RMS volume.

Parameters

- **Input velocity volume** — the volume the model is built from. The list contains the 2D lines and 3D cubes of the project whose data type is Velocity, in both domains; it stays empty when the project holds no such data.
- **Velocity type** — what the selected volume stores: **RMS velocity** (default), **Interval velocity** or **Average velocity**. The conversion follows this setting, so a volume declared with the wrong type produces wrong velocities.
- **Datum** — elevation reference of the model (editable).
- **Max depth** — lower bound of the model (default 4000 m).
- **Depth step** — vertical sampling of the model (default 10 m).
- **Vertical smoothing window** — moving average applied vertically to the resulting model, in samples. 0 (default) leaves the model unsmoothed.
- **Horizontal smoothing window** — lateral smoothing along the time horizons of the project, in traces (default 20); 0 switches it off. It applies to a time-domain RMS volume, the other combinations are not smoothed laterally.



How the volume is converted

- **Time volume, RMS velocity** — Dix conversion to interval velocity and resampling to depth, with the optional smoothing along horizons.
- **Time volume, Interval velocity** — the interval velocities are converted from time to depth.
- **Time volume, Average velocity** — the average velocities are turned into interval velocities first, then converted from time to depth.

© 2026 Geomage 2003 LTD

- **Depth volume, RMS velocity** — Dix conversion to interval velocity in depth.
- **Depth volume, Interval velocity** — taken as it is, no conversion is applied.
- **Depth volume, Average velocity** — the average velocities are turned into interval velocities.

A depth volume keeps its own depth range and sampling, so **Max depth** and **Depth step** shape the model only when the input is in the time domain. The vertical smoothing window is applied to the resulting depth model in both cases.

Workflow

1. Select **Velocity volume** under **Velocity models** in the velocity model wizard.
2. Choose the volume in **Input velocity volume** and set **Velocity type** to what that volume contains.
3. Set **Datum**, **Max depth** and **Depth step**.
4. Enter the smoothing windows the model needs, or leave them at their defaults.
5. Run the model. The selected volume is converted to interval velocity in depth, smoothed as requested, and written as the new velocity model.

Output

Interval velocity model in depth, covering the geometry of the selected volume and referenced to the datum you entered.

When to use

- Areas with good seismic velocity coverage (for example fresh processing results) and limited well data.
- Starting model for later refinement with well- or marker-based methods.
- High-lateral-resolution use cases where seismic velocities capture features unavailable from sparse wells.

For well-tied models, combine this initial cube with Checkshot + markers + horizons 670 in a second pass.

See Also

- Setting up the velocity model 660
- Checkshot + markers + horizons 670
- Velocity model QC 675

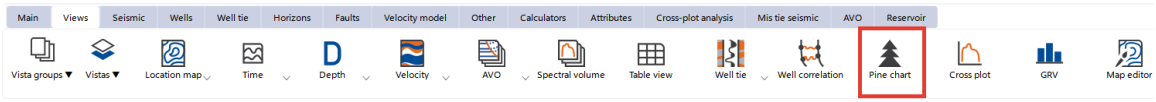
© 2026 Geomage 2003 LTD

9.6.2 Velocity model QC

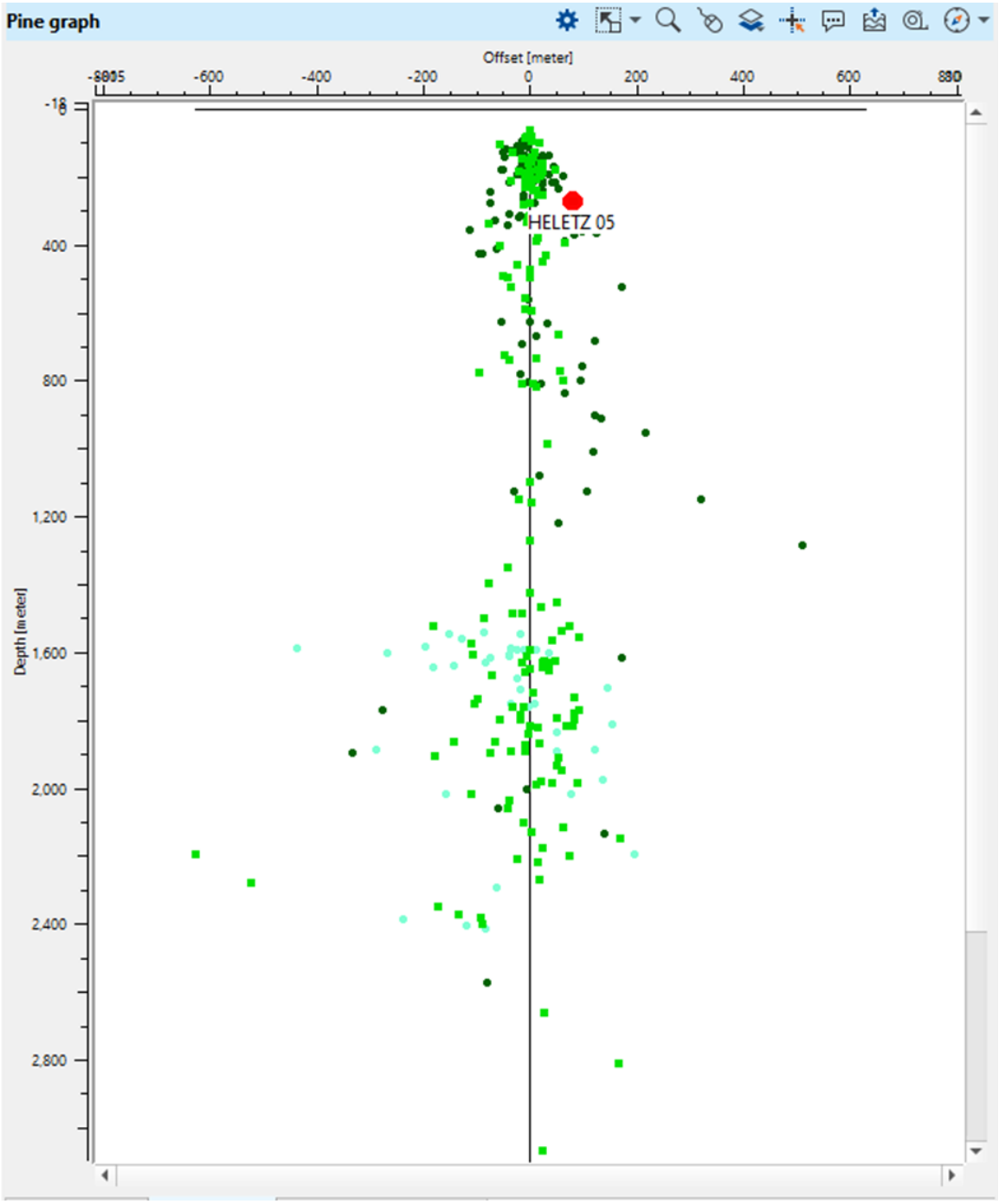
To access the **Velocity model QC** tool navigate to the Views and

The Pine chart tool allows users to visualize and compare the velocity model with well data in a two-dimensional space. The chart is structured as follows:

- **X-Axis (Offset)**: Represents the distance from
- **Y-Axis (Depth)**: Indicates the depth of the geological layers, which is essential for correlating seismic data with subsurface features.



© 2026 Geomage 2003 LTD



To compare depth horizons in the **3D view** using different velocity models, you can convert time horizons to depth maps based on the currently active velocity model. Depth maps are stored in the **Maps** folder in the **Data Manager**.

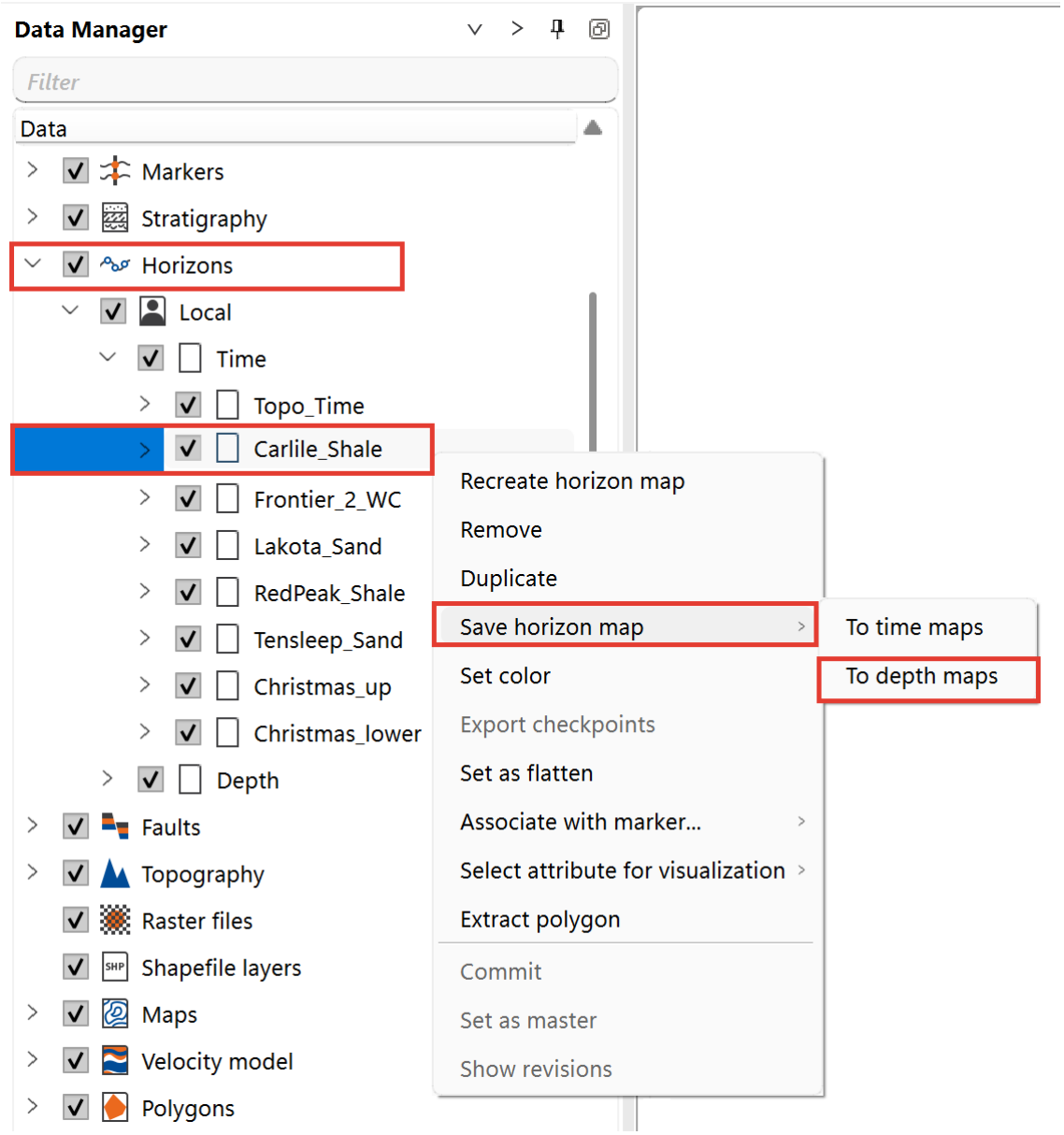
To perform this comparison:

1. Set the active velocity model.
2. Convert the time horizon to a depth map.

© 2026 Geomage 2003 LTD

3. Change the active velocity model.
4. Repeat the conversion process for the same time horizon.

This workflow allows you to visualize and compare depth variations resulting from different velocity models directly in the 3D view 244.



© 2026 Geomage 2003 LTD

9.6.3 Import velocity model

Import velocity model loads a depth velocity model stored in a **SEG-Y** file (`.sgy` / `.segy`) into a g-Space project. The velocity data is read through the standard SEG-Y reader, but the command lives on its own velocity menu rather than the seismic import.

Purpose

Bring an externally prepared depth velocity model into g-Space as a new velocity model. Each imported trace becomes a velocity function in depth, so the file is interpreted in the depth domain as post-stack velocity data.

Launching the import

In the **Data manager**, right-click the **Velocity model** root node and choose **Import velocity model** from its context menu.

Step-by-step

1. In the **Import depth velocity** file dialog, select one or more `.sgy` or `.segy` files (filter `Data files (*.sgy *.segy)`).
2. Set the **Trace header format** so that the SEG-Y reader maps trace coordinates and geometry correctly.
3. Provide the coordinate reference (EPSG) information requested by the import dialog so the velocity traces are positioned in the project coordinate system.
4. Confirm to read the file. g-Space snaps traces to the nearest existing bins, builds a velocity gather in the depth domain, and derives the model datum from the data.
5. A new velocity model is created and set as current in the **Velocity model** tree, ready for use.

Notes

- The file is always interpreted as a depth-domain, post-stack velocity model, so the values in the traces should be velocities.
- Because the standard SEG-Y reader is used, the same trace-header and coordinate-reference settings that apply to seismic import apply here.

© 2026 Geomage 2003 LTD

- Imported traces are matched to the project geometry by snapping each trace to the nearest bin, so a compatible survey geometry should already exist in the project.

See Also

Seismic Data Formats 511 | Export velocity model 679 | Velocity model 658

9.6.4 Export velocity model

Export velocity model writes a depth velocity model from a g-Space project to a **SEG-Y** file (`.sgy` / `.segy`), so the model can be reused in g-Space or in other software.

Purpose

Save an existing depth velocity model as a SEG-Y file. Each velocity function is written out as a trace in the depth domain, preserving the model's geometry and sampling.

Launching the export

In the **Data manager**, expand the **Velocity model** node, right-click the velocity model you want to save and choose **Export to Seg-Y file**.

Step-by-step

1. Right-click the target velocity model and select **Export to Seg-Y file**.
2. In the save dialog, choose a location and file name; the filter is `Seg-Y file (*.sgy)`.
3. Confirm to write the file. g-Space exports the model gather with its depth sample interval into the new SEG-Y file.

Notes

- The command is available only on an individual velocity model entry, not on the Velocity model root node.
- The model is exported in the depth domain, matching the way it was imported or built.

© 2026 Geomage 2003 LTD

- The resulting SEG-Y file can be read back with Import velocity model.

See Also

Seismic Data Formats 511 | Import velocity model 678 | Velocity model 658

9.7 Seismic attributes

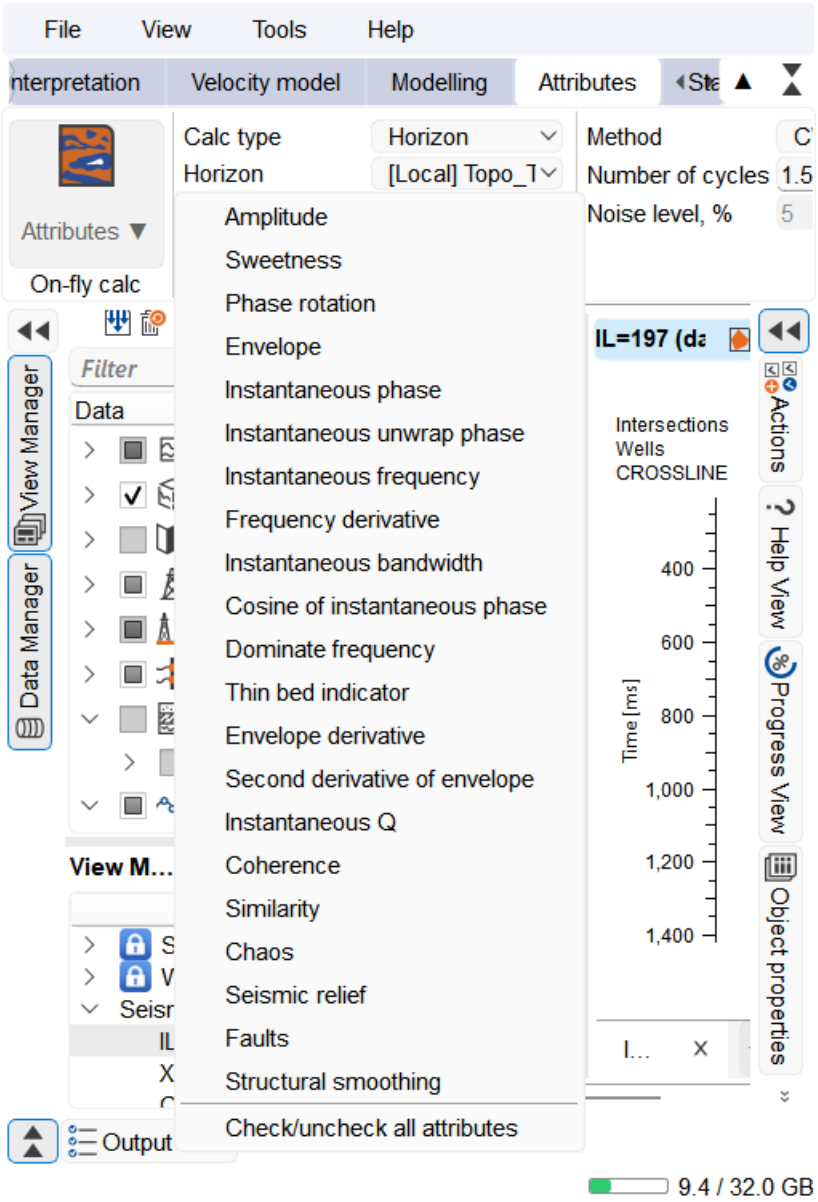
In g-Space, seismic attributes are tools that extract additional insights from seismic data, offering detailed information about the subsurface.
Seismic attributes can be computed on 2D and 3D data, providing flexibility for seismic volume analysis, horizon interpretation, and fault detection. The attribute settings are configured in the Attributes bar 105.

First, it's important to note that g-Space offers two methods for calculating attributes: a quick **On-fly calculation** and a more comprehensive full **Calculate attributes** for seismic volumes, including spectral decomposition.

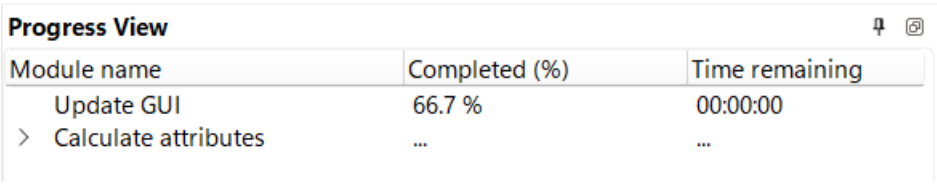
How to Generate On-fly calculation Seismic Attributes

1. Click on the **Attributes** icon in the toolbar.
2. Set the time window if applicable to the attribute.
3. Select the seismic attributes you wish to calculate from the drop-down list. You can choose one or multiple attributes simultaneously.

© 2026 Geomage 2003 LTD



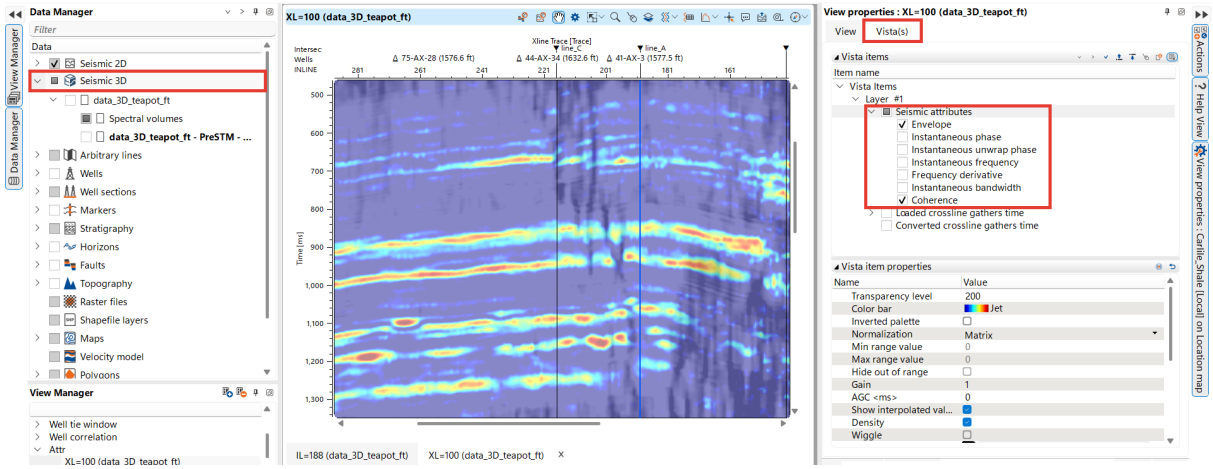
The process takes seconds, the progress can be seen on the right side in the Progress View.



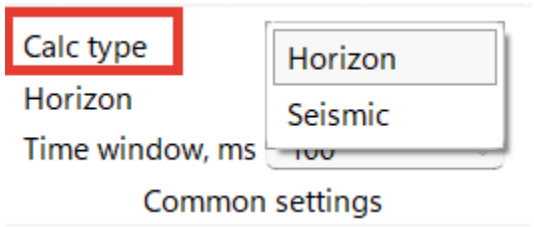
The calculated attribute will be accessible in any seismic view. In the example below, it is shown in the Crossline view. In the **Visual Settings** panel, users can select one or multiple attributes to

© 2026 Geomage 2003 LTD

display, customize the color palette, adjust transparency, and modify other visualization settings.



How to Generate Volume based Seismic Attributes



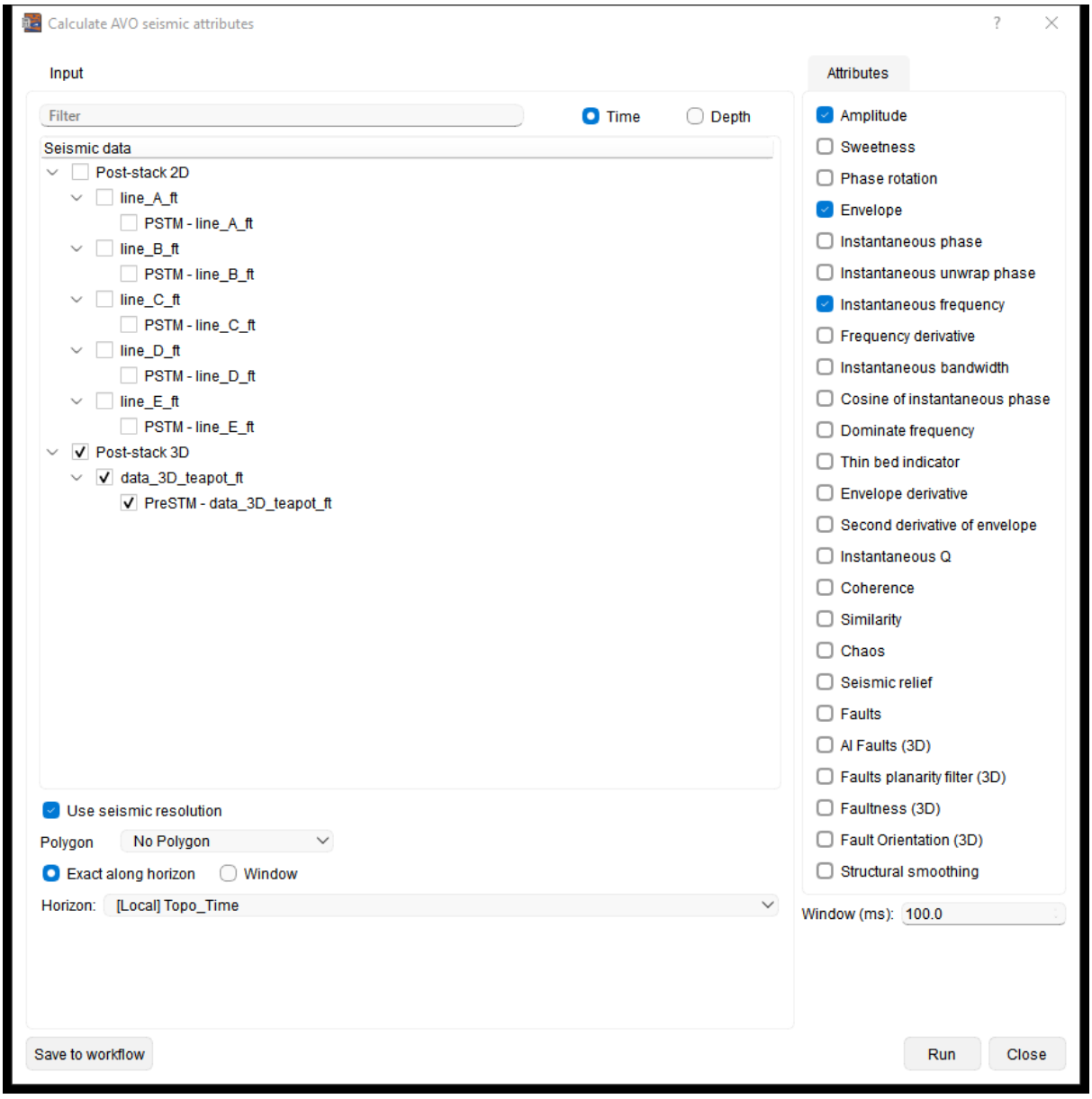
First, navigate to the **Attributes** bar and, in the **Common Settings** tab, select the calculation type — **Horizon** or **Seismic**. Then, go to the **Execute** tab and click **Calculate Attributes**. The **Calculate AVO seismic attributes** wizard opens, where you can choose seismic data and attributes for calculation.
Click **Run** to calculate immediately with the current settings, **Save to workflow** to add those settings to the current workflow 535 without running the calculation, or **Close** to dismiss the wizard. When you open the saved task from the workflow, its settings are restored; use **Save** to update the task or **Cancel** to close without updating it.

There are two options — use **Exact along horizon** to calculate on a selected surface, or **Window** to calculate over a specified interval.

© 2026 Geomage 2003 LTD

Both options share two further settings. **Use seismic resolution** is selected by default and is available only when exactly one 3D volume is selected; when it is on, the attribute map is written directly on the seismic bin grid instead of being interpolated. **Polygon** lists the polygons of the project and is set to **No Polygon** by default; selecting a polygon limits the resulting attribute map to that area, both for the bin-grid map and for the interpolated map.

For **Exact along horizon**, select the target surface from the **Horizon** list. In **Time** mode, the list includes time horizons and time maps; in **Depth** mode, it includes depth horizons.



The **Window** option allows users to calculate seismic attributes over a specified time window relative to a selected horizon or time interval.

© 2026 Geomage 2003 LTD

There are three options available:

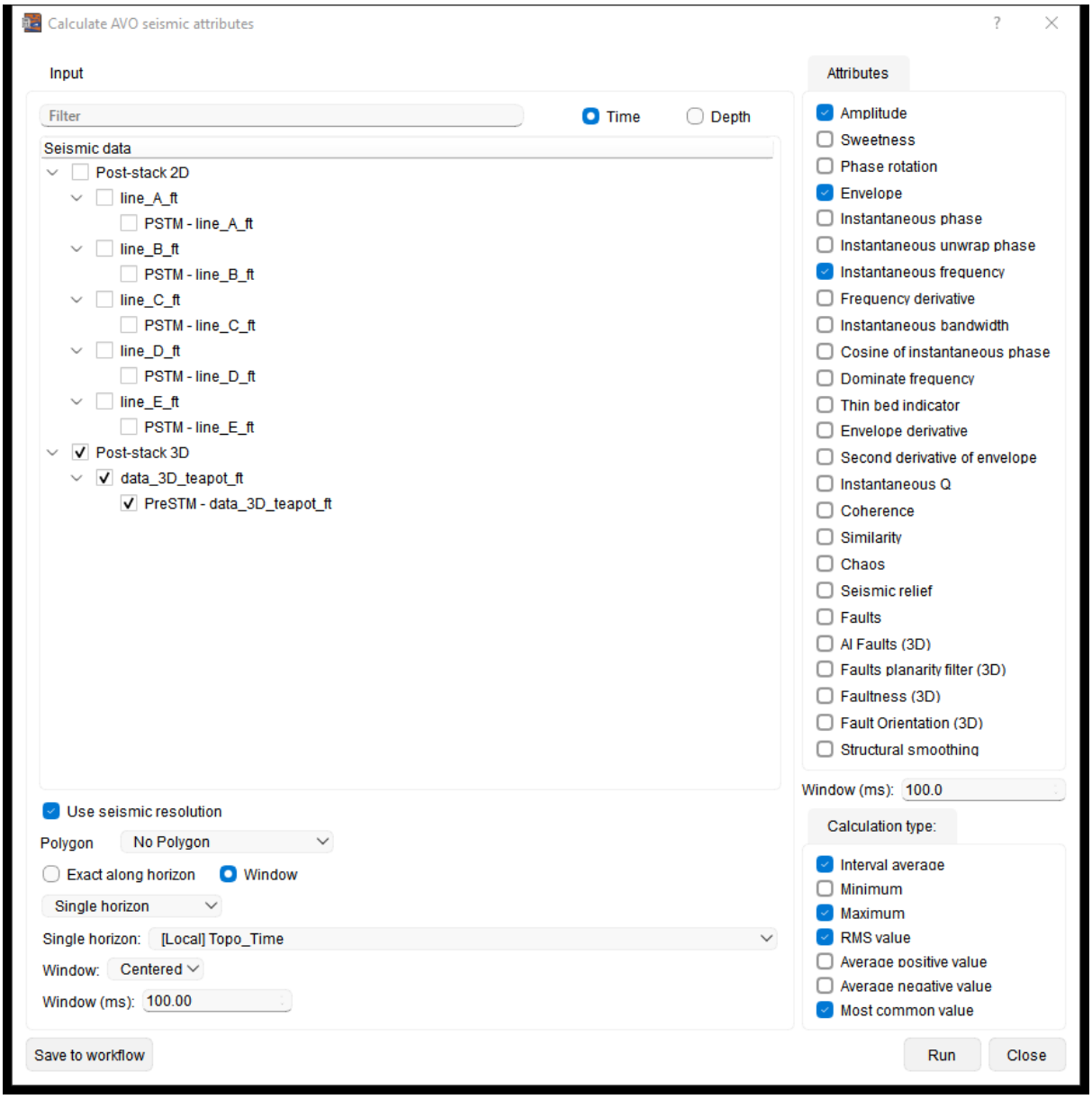
Between time: attributes are calculated over a specific time interval. Users can specify the start and end times (in milliseconds) within the seismic data.

Between horizons: the option calculates attributes between two selected surfaces. Choose the start and end surfaces from the drop-down menus; in **Time** mode, the lists include time maps as well as time horizons.

Single horizon: calculates attributes around one selected surface. Choose the surface from the drop-down list; in **Time** mode, the list includes time maps as well as time horizons. The **Window** (in milliseconds) determines the time range used to calculate the attributes. The window can be positioned **Above**, **Below**, or **Centered** (with the horizon in the middle, equally above and below) for attribute calculation.

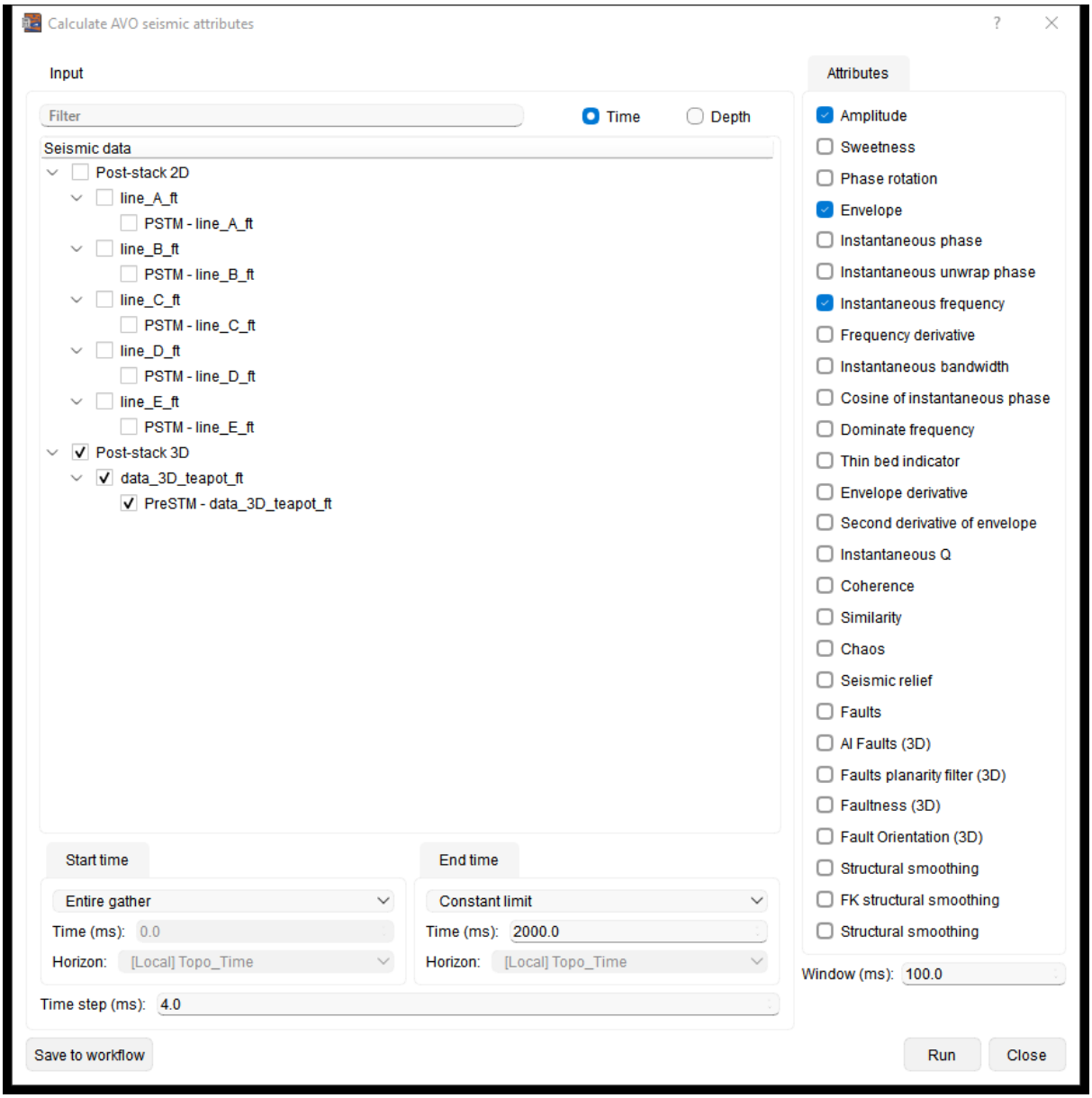
Users can select different statistical operations such as **Interval average**, **RMS value**, **Maximum**, **Minimum**, **Average positive/negative value**, or **Most common value** to be applied within the specified window.

© 2026 Geomage 2003 LTD



For the **Seismic** calculation type, use the **Start time** and **End time** controls to calculate the entire gather or define a **Constant limit** or **Horizon limit**. In **Time** mode, **Horizon limit** lists time horizons and time maps; in **Depth** mode, it lists depth horizons. Set the **Time step** or **Depth step** and click **Run**. Monitor the calculation in the **Progress View**.

© 2026 Geomage 2003 LTD



Attribute parameters in the SEG-Y text header:

For the **Seismic** calculation type each attribute is written to its own SEG-Y file, and the settings used for the calculation are stored in the text header of that file, so a volume can still be traced back to the parameters that produced it after it is reopened or passed on to somebody else. Every file records the attribute name, the time step and the date the file was written (UTC, in YYYY-MM-DD form), plus the time window for attributes that are calculated over a window.

Attributes that have settings of their own also record those settings:

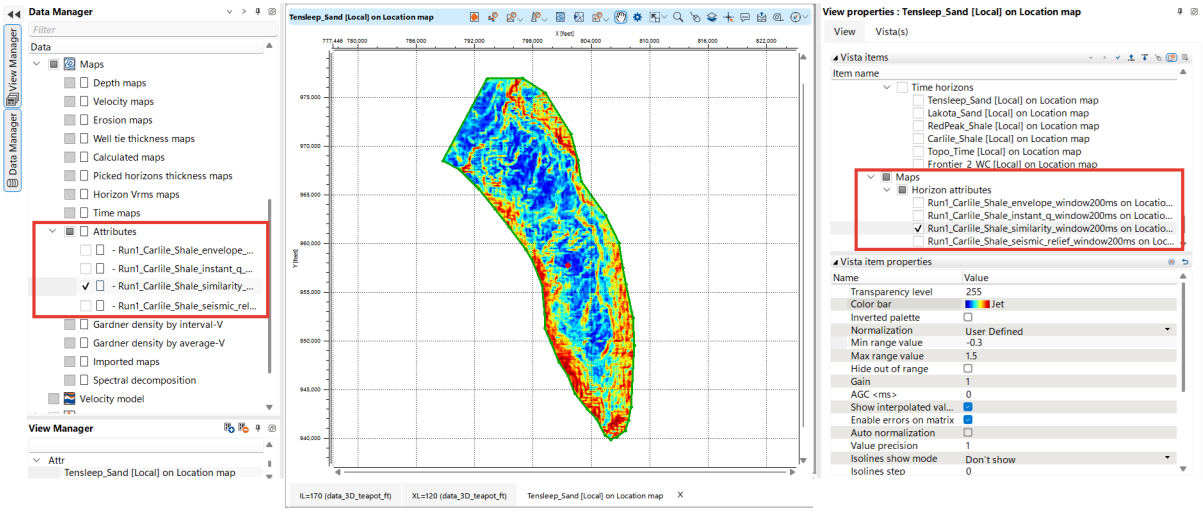
© 2026 Geomage 2003 LTD

- **AI Faults (3D)** — inference patch size
- **Faults planarity filter (3D)** — volume size reduction, edge detection strength and fault continuity
- **FK structural smoothing** — horizontal sliding window, time window, time taper window, min and max frequency, and the number of Eigen values
- **Structural smoothing** — sigma inline, sigma crossline, sigma samples and lambda
- **Seismic relief** — vertical window and pow factor

The entries are placed in the spare lines of the standard 40-line text header (C33, C36, C37, C38 and C40), so the trace-header byte map on the lines above them is preserved; when an attribute has more parameters than there are spare lines, two of them share a line. To read the header, use the **Text header** view of the seismic data 228 workspace.

View Attributes:

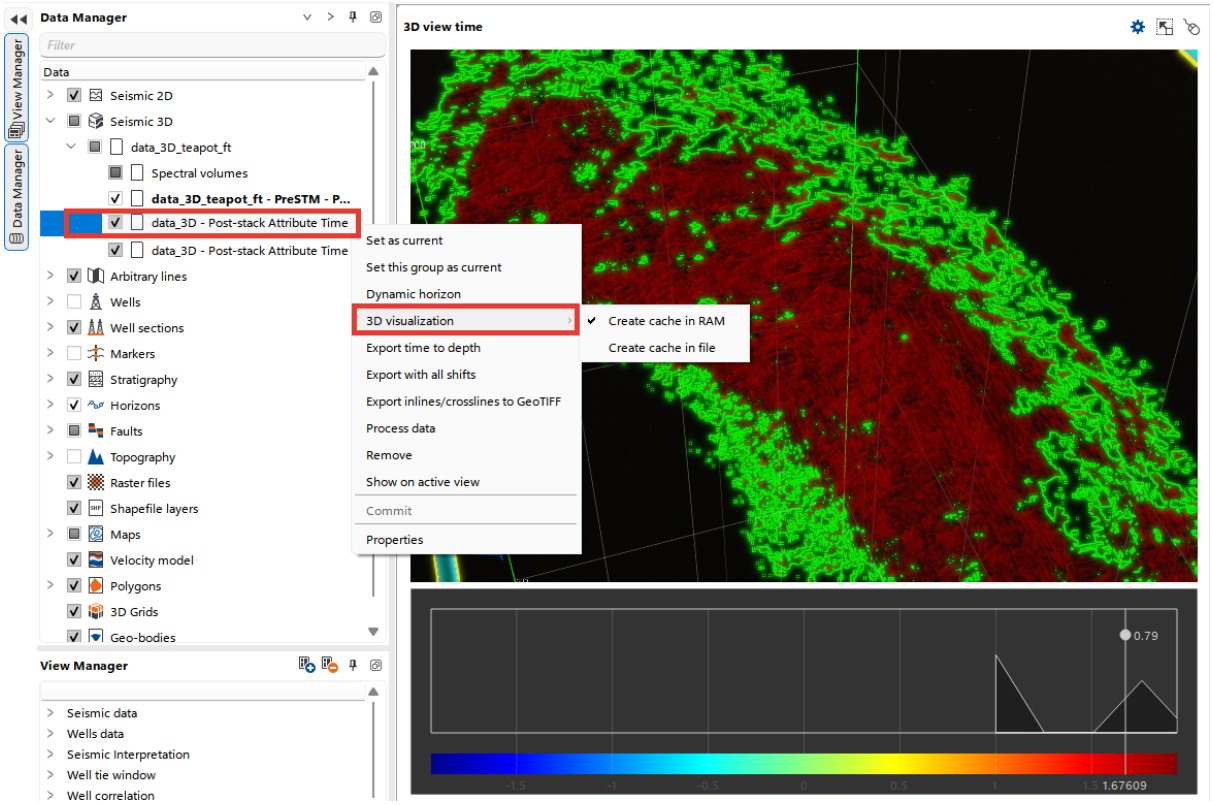
To view the attributes, user should go to **Maps** section in the **Data Manager.** Expand the **Maps** section and click the **Attributes** section.



All the calculated attributes should be visible on the Location map.

The user can visualize the calculated attribute volumes as any 3D seismic data 599. To do this, you need to create a cache in RAM or in a file by right-clicking on the calculated attribute volume in the Data Manager 124. A special volume palette is available in 3D view to customize the visualization of attribute volumes. Adjust the image by dragging the slider along the horizontal and vertical scale.

© 2026 Geomage 2003 LTD

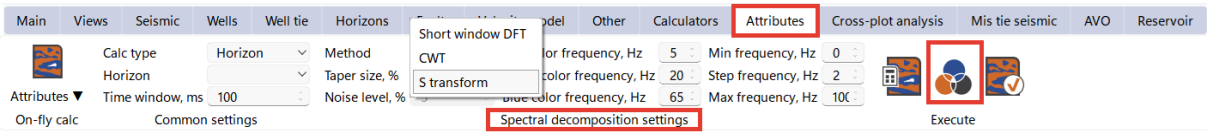


9.7.1 Spectral decomposition

Spectral decomposition is a technique in seismic interpretation used to analyze the frequency content of seismic data. By using spectral decomposition, we can visualize how different frequency bands respond to subsurface features, enhancing reservoir characterization.

How to Perform Spectral Decomposition

First, open the **Attributes Bar** 105 where you can configure your spectral decomposition settings. From here, you'll select the horizon or seismic data and the time window for which you want to perform the spectral decomposition.



© 2026 Geomage 2003 LTD

g-Space offers three calculation Methods for spectral decomposition. Choose the one that fits your geological objectives. These methods vary in terms of how they handle time windows and frequency ranges, which may affect the resolution and interpretability of the results.

1. ***Short Window DFT (Discrete Fourier Transform)***

The Short Window DFT method applies a fixed-length time window to the seismic trace and performs a Discrete Fourier Transform within that window. This approach offers a balance between time and frequency resolution but may lose precision for rapidly changing signals due to its fixed window size. Ideal for stable, continuous stratigraphy or thick-bedded sequences where uniform frequency content is expected over short intervals.

2. ***CWT (Continuous Wavelet Transform)***

The CWT method uses wavelets of varying scales to analyze the seismic signal. Unlike DFT, the CWT provides a multi-resolution analysis by adapting the time window size to the frequency — wider for low frequencies and narrower for high frequencies. Best for resolving thin beds, complex stratigraphy, and fault zones where frequency content changes rapidly over short time periods.

3. ***S-Transform***

The S-Transform combines elements of both the Short Window DFT and CWT. It uses a variable window length, similar to CWT, but retains a direct relationship with Fourier analysis. The S-Transform provides both good time and frequency localization, making it suitable for detecting frequency anomalies at different scales in complex geological environments. Effective for detecting tuning thickness, complex channel systems, and subtle reservoir heterogeneities due to its balance of time and frequency resolution.

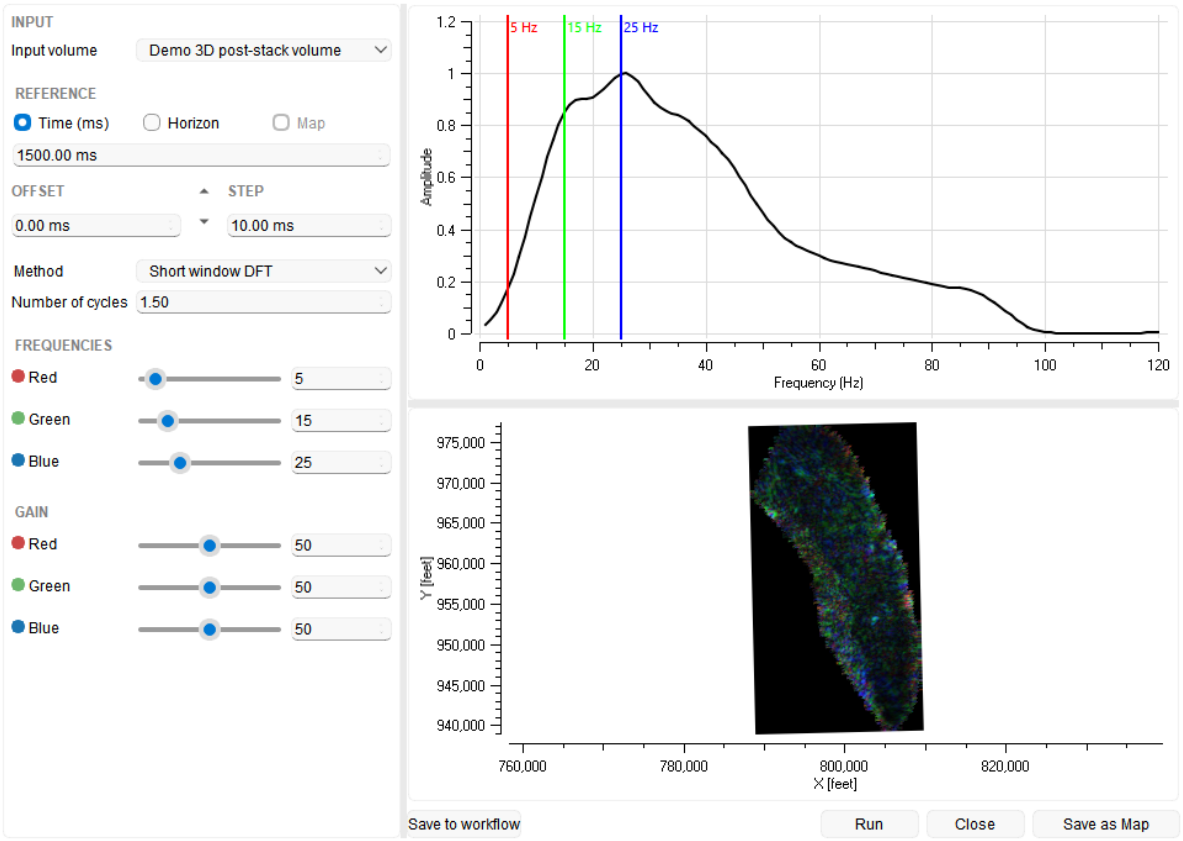
Set Frequency and Window Parameters

After choosing your method, you need to set several key parameters:

- **Number of Cycles**: A dimensionless control of the analysis window length (window = cycles / frequency). Larger values give finer frequency resolution, while smaller values give finer time resolution.
- **Noise Level (%)**: This determines how much noise reduction will be applied during the decomposition process.
- **Color Frequencies**: Assign specific frequencies to each color (Red, Green, Blue) for visual blending. This helps to emphasize different frequency components in the final output.
- **Min, Max, and Step Frequencies**: Define the range and increment of frequencies analyzed. The **Min** frequency represents the lowest frequency to analyze, the **Max** frequency is the highest, and the **Step** frequency defines the intervals between frequency bands.

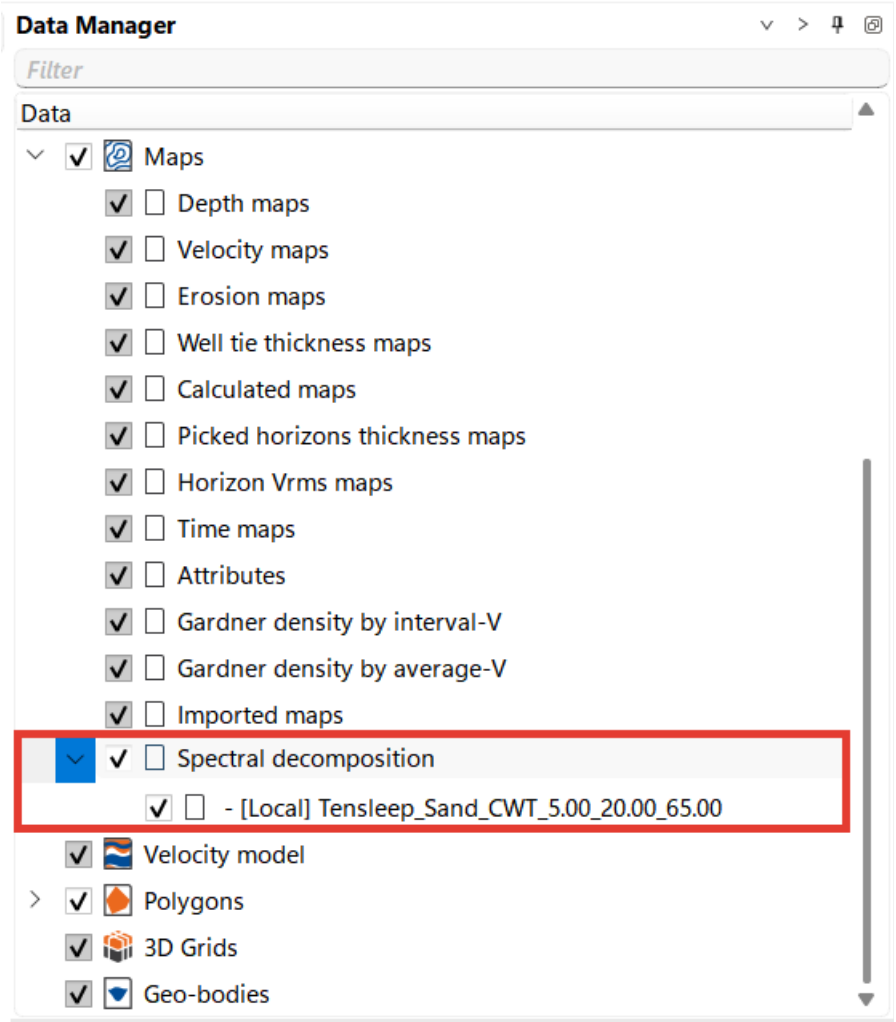
© 2026 Geomage 2003 LTD

Once all parameters are set, click **Spectral decomposition** in the **Execute** group of the Attributes Bar. In the wizard, choose the input volume and reference, then adjust the decomposition settings. **Save to workflow** stores the selected input and current settings as a task in the active workflow 535 without running the calculation and leaves the wizard open; when the workflow reaches the task, it calculates the decomposition and saves the map. **Run** calculates and saves a map immediately and leaves the wizard open. **Save as Map** saves the current preview, and **Close** closes the wizard. When a saved task is reopened from the workflow, its settings are restored and the footer shows **Save** and **Cancel**. **Save** updates the task; **Cancel** closes it without changing the task.



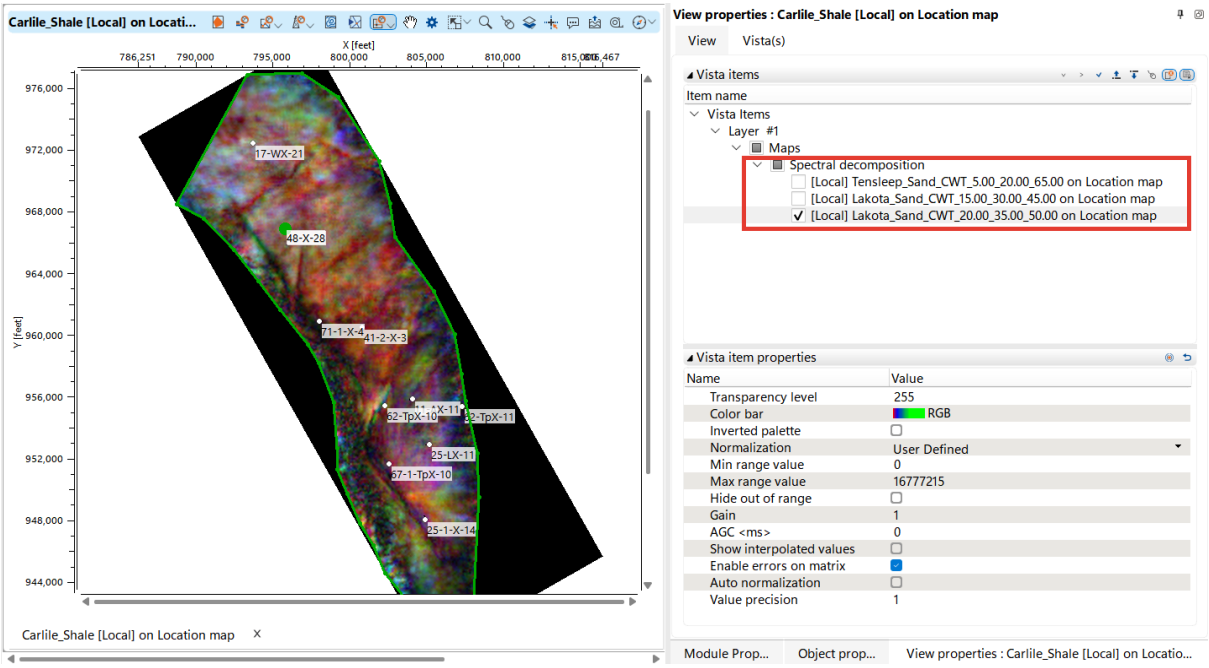
After processing, a new map folder will appear in the **Data Manager** under the **Maps** folder. This map represents the spectral decomposition output and can be visualized on the location map. The method and the frequencies used in the calculation will be automatically written into the map name.

© 2026 Geomage 2003 LTD



An example of a spectral decomposition map visualization is shown below. Users can customize the visualization in **Visual Settings** by adjusting the color palette and other available options.

© 2026 Geomage 2003 LTD

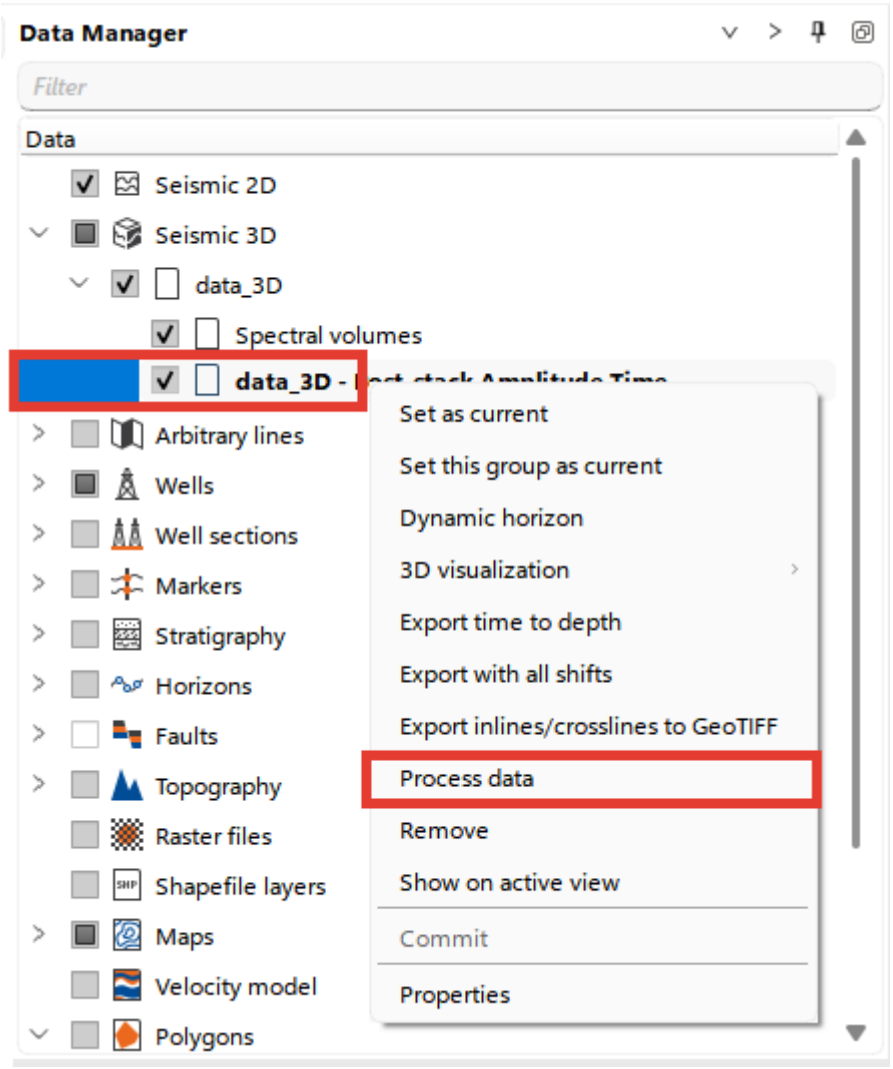


9.8 Gather conditioning

g-Space allows performing seismic data processing by connecting the g-Navigator package via the g-Space Flow module.

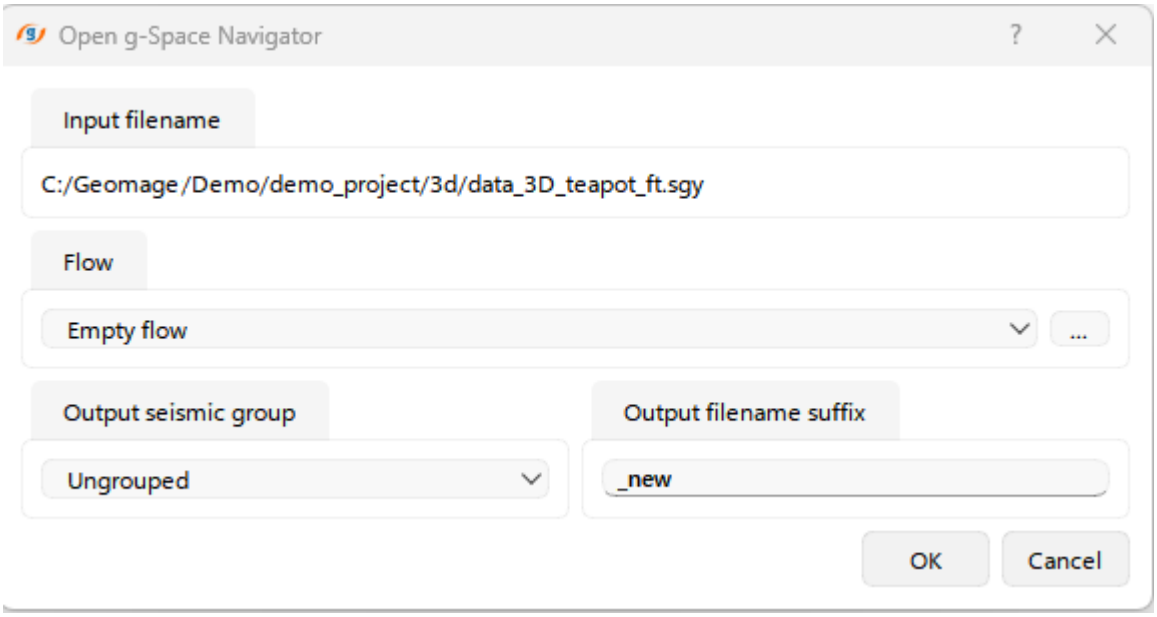
Navigate to **Data Manager**.
Right-click on the seismic data and choose **Process Data**.

© 2026 Geomage 2003 LTD



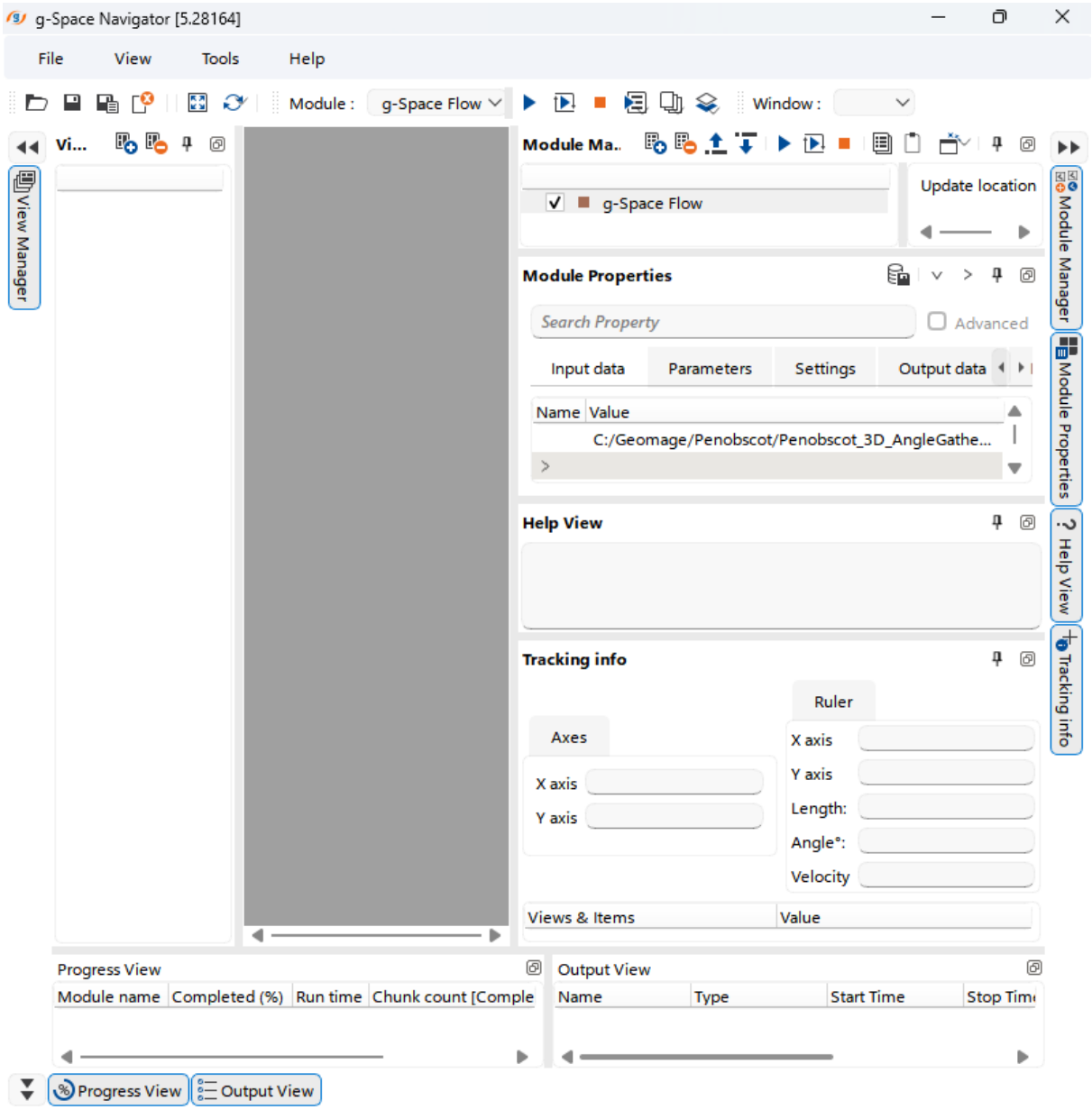
The **Open g-Space Navigator** window will appear. Fill in the display names and click **OK**.

© 2026 Geomage 2003 LTD



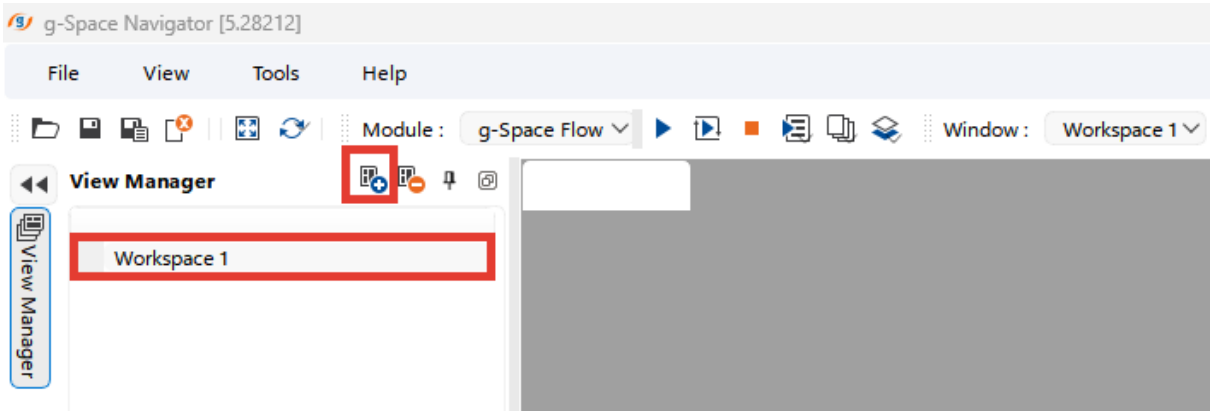
The **g-Space Navigator** module will open.

© 2026 Geomage 2003 LTD

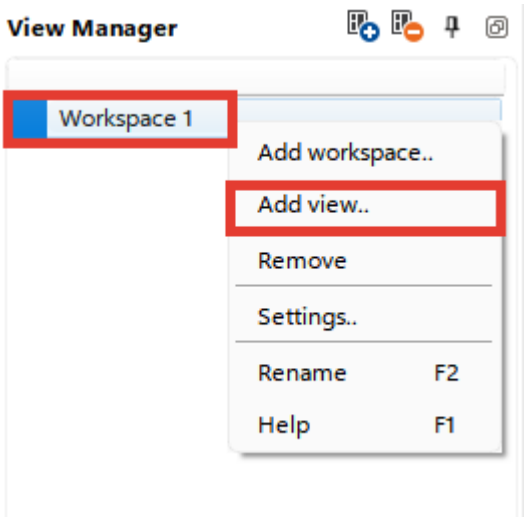


In the **Navigator module**, click the **+** button and create a new workspace.
Select the newly created workspace to proceed.

© 2026 Geomage 2003 LTD

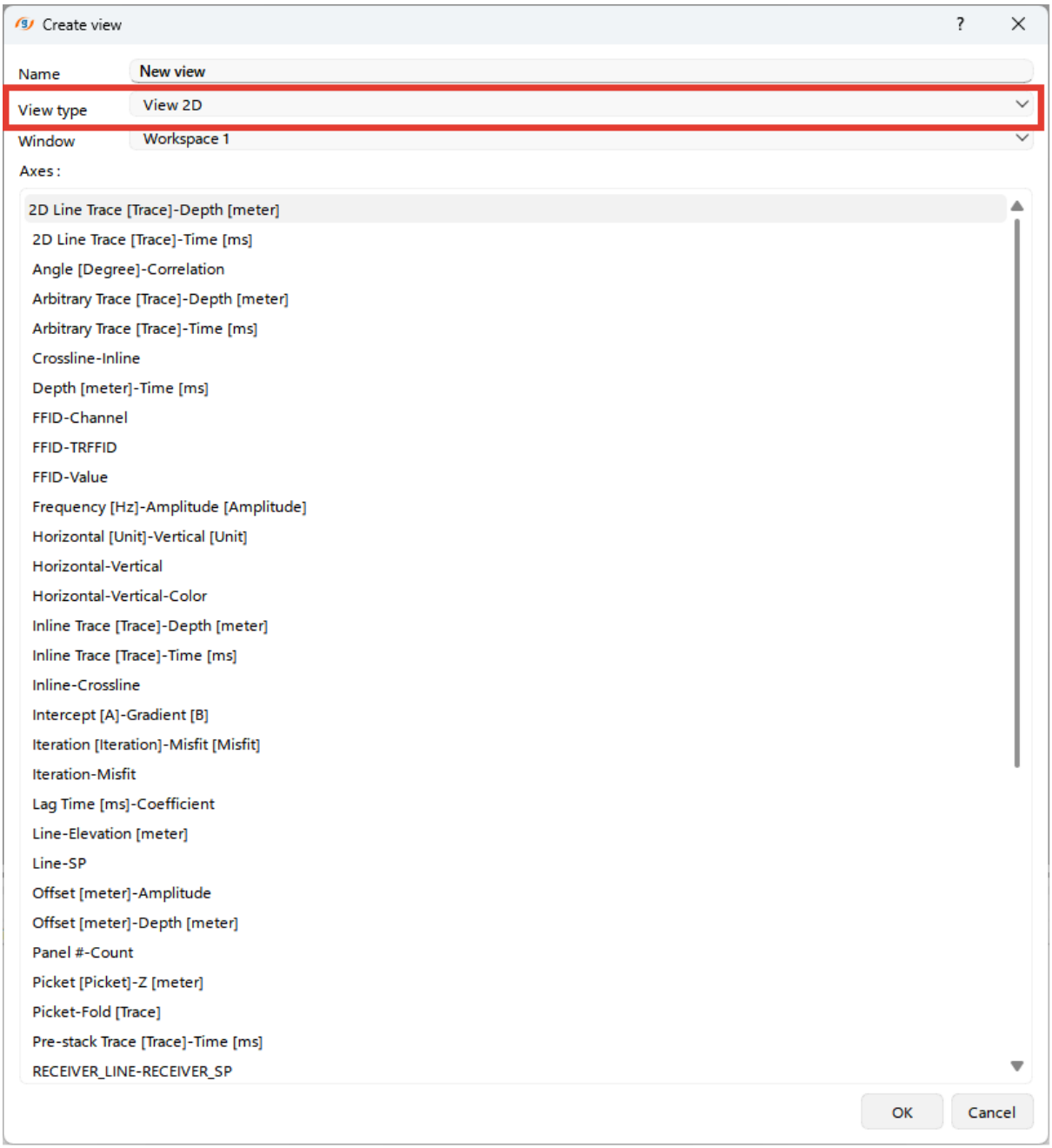


Then create a new view for your data.



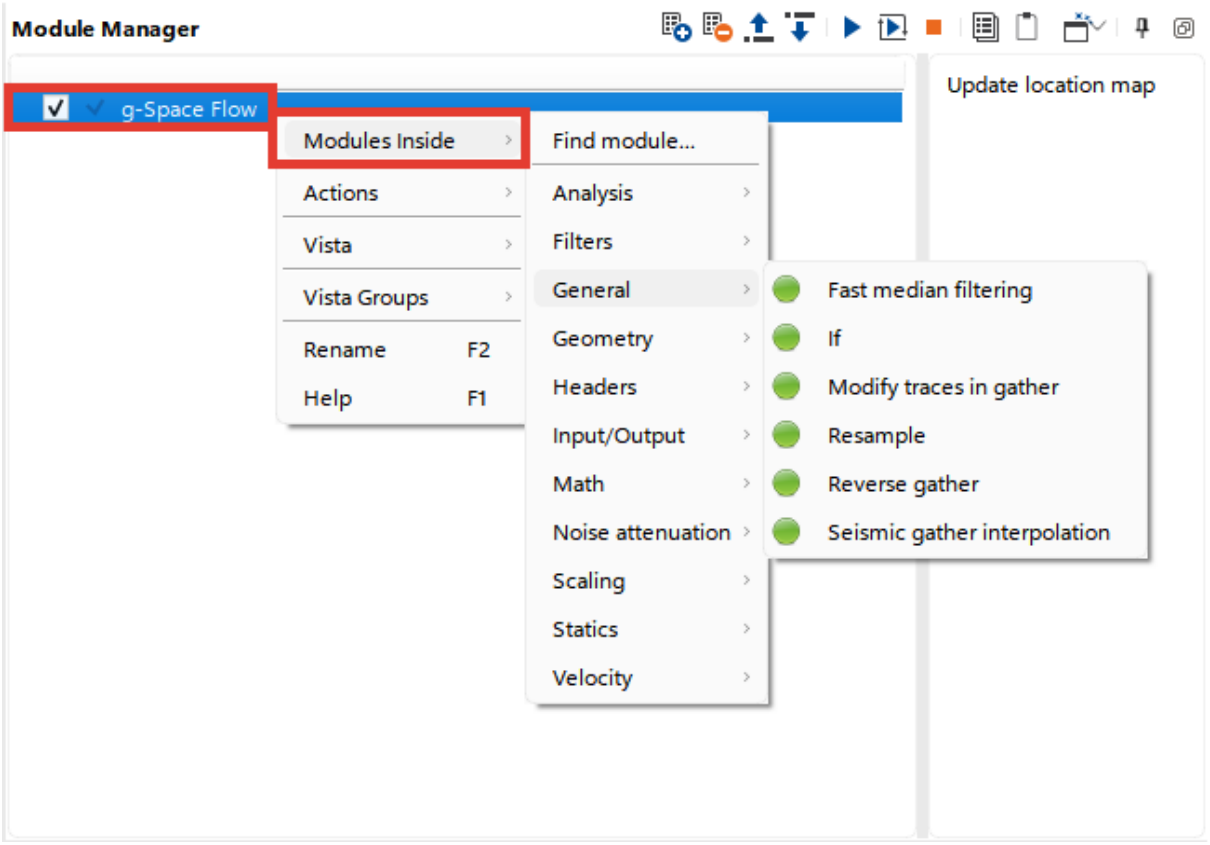
Choose **View type** and a particular pair of **Axes**

© 2026 Geomage 2003 LTD

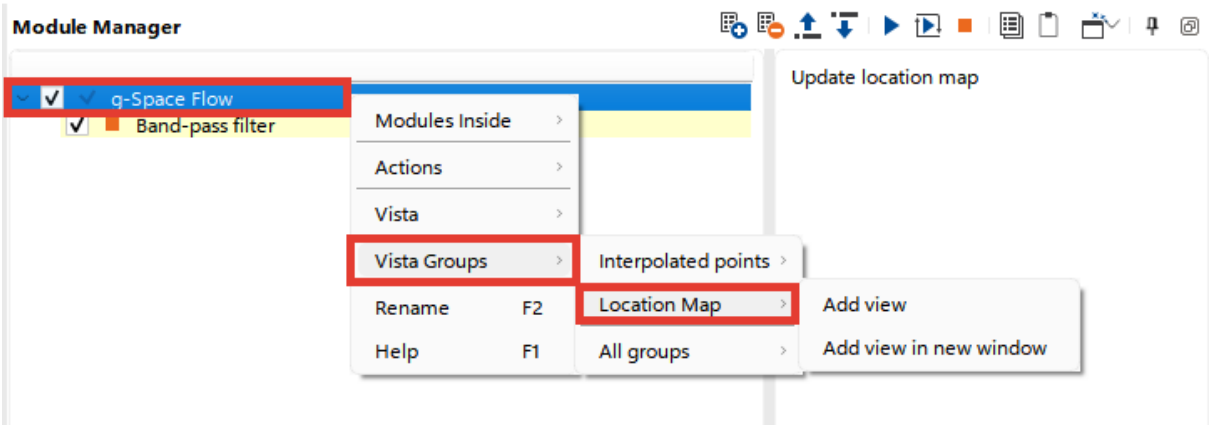


Then navigate to the **Module Manager**, right-click and select the desired modules

© 2026 Geomage 2003 LTD

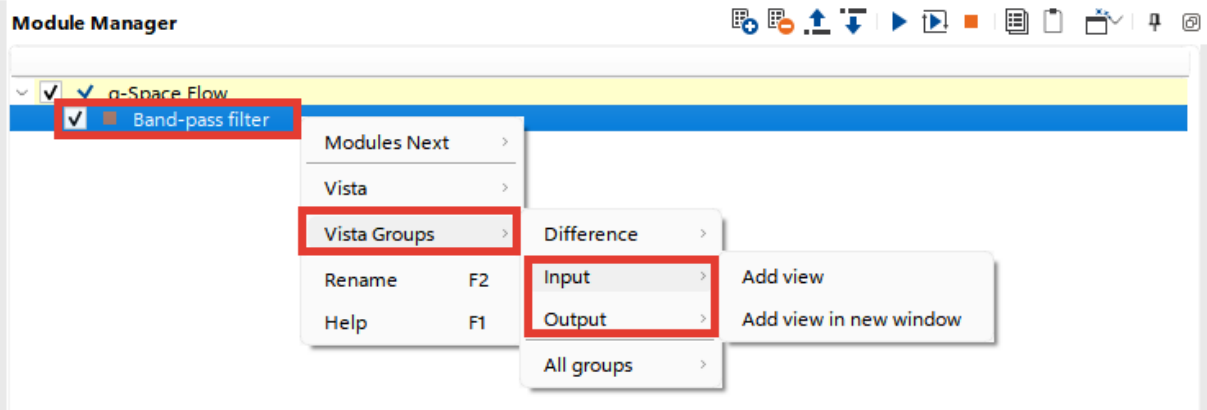


After that click on the **g-Space flow** and add the **Location Map view** that will allow you to select and visualize the data you are working with



Click on the selected modules to display the Input and Output views for quality control.

© 2026 Geomage 2003 LTD

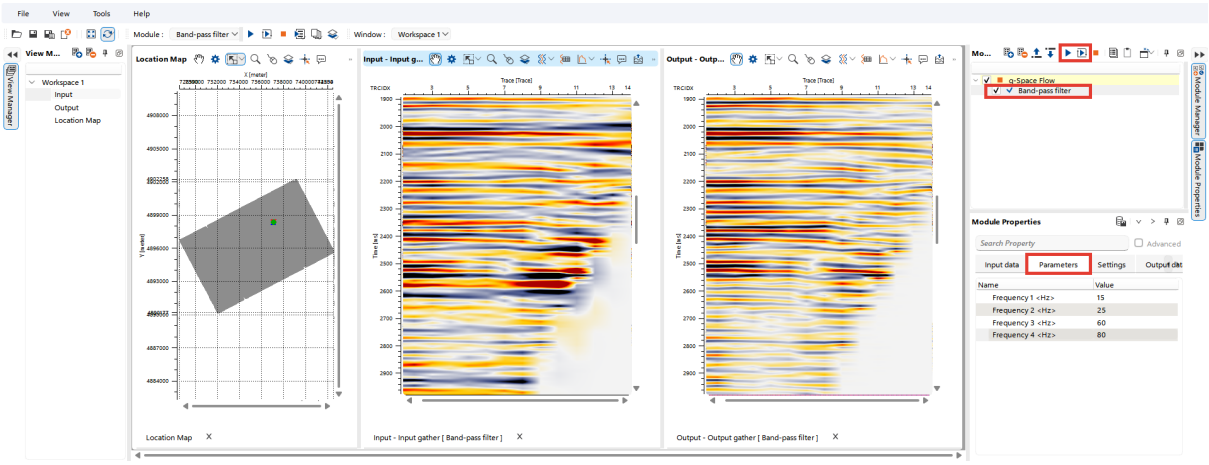


The configured interface displays the **Parameters** section, where you can set parameters for the selected module. Results appear instantly in the configured Input and Output views for a selected bin active on the **Location Map**.

Select the **g-Space Flow** module itself to control how the result is written to the output file. Its **Parameters** tab contains the **Saving mode** option:

append (default) writes only the traces that the flow produces, so modules that reduce the amount of data, such as **Stack**, give a compact output volume.
direct writes the traces in the sorting order of the input data, so the output keeps the size of the input gathers.

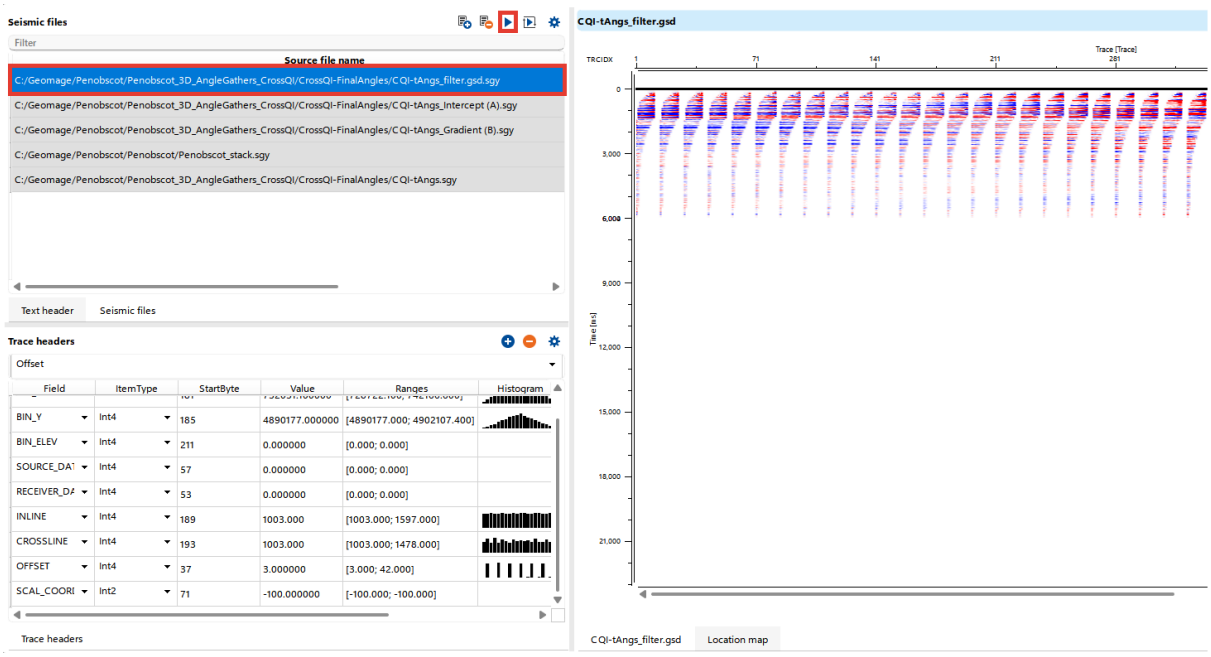
Use the **Execute** or **Execute All Modules** buttons to process the data.



For details on seismic data processing functionality, refer to the [g-Platform]

© 2026 Geomage 2003 LTD

After obtaining the results, return to g-Space. In the Seismic Data workspace, you will find your files. Upload them if needed.

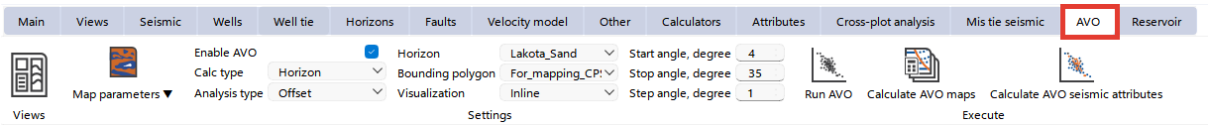


Note

The default file name for upload is limited to 7 characters. If you add a suffix to the original file name during processing, ensure the full name fits within the 7-character limit before uploading.

9.9 AVO

AVO (Amplitude Variation with Offset) analysis in g-Space allows users to measure rock properties and differentiate between various subsurface materials based on how seismic amplitude changes with offset. Through this analysis, users can generate key parameters like the **Intercept**, **Gradient**, and their products to assess the subsurface.



In the AVO tab, there are three main sections:

© 2026 Geomage 2003 LTD

1. **Map Parameters**: Where the user can select different AVO attributes like Intercept, Gradient, or their products.
2. **Settings**: Options to enable or disable AVO, choose 2D/3D visualization, and select available horizons.
3. **Execute**: The section where the user runs the AVO calculations.

A full description of the functionality is available in the AVO bar 119

To visualize the output(s) of AVO, the user can open **Views** in the AVO bar or AVO views 272 in the **Views** bar.

AVO Analysis procedure

To conduct AVO analysis, you will need:

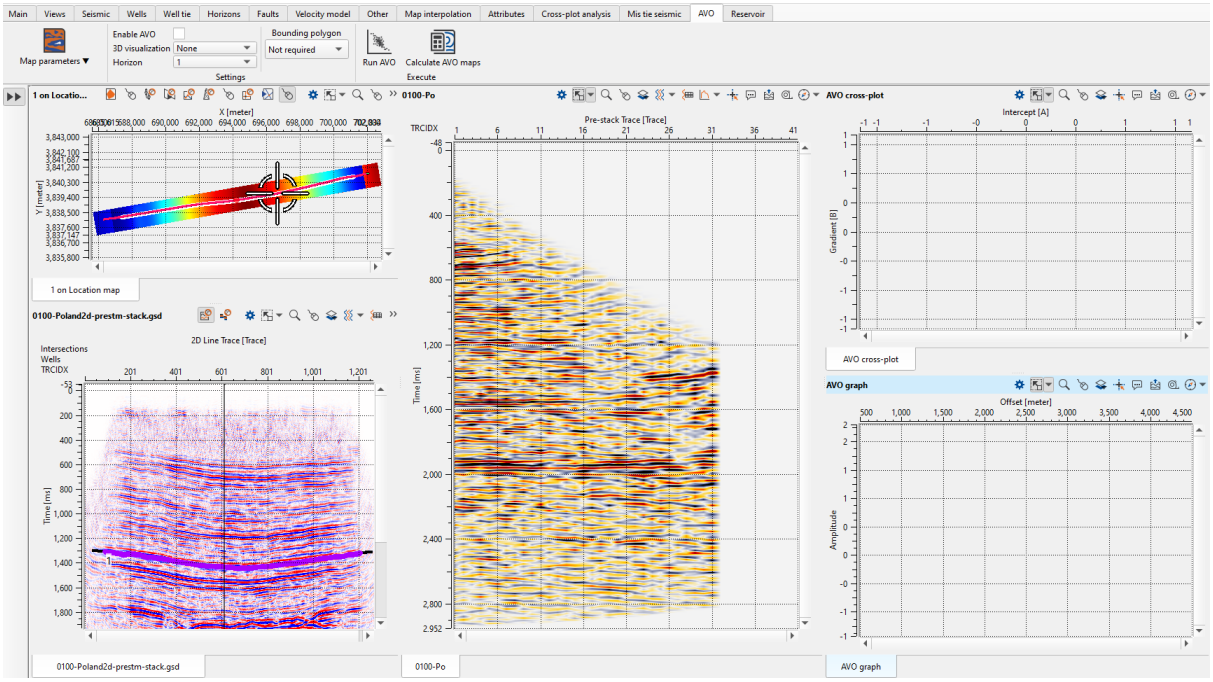
- Migrated gathers
- Interpreted horizons

Follow these steps for AVO analysis:

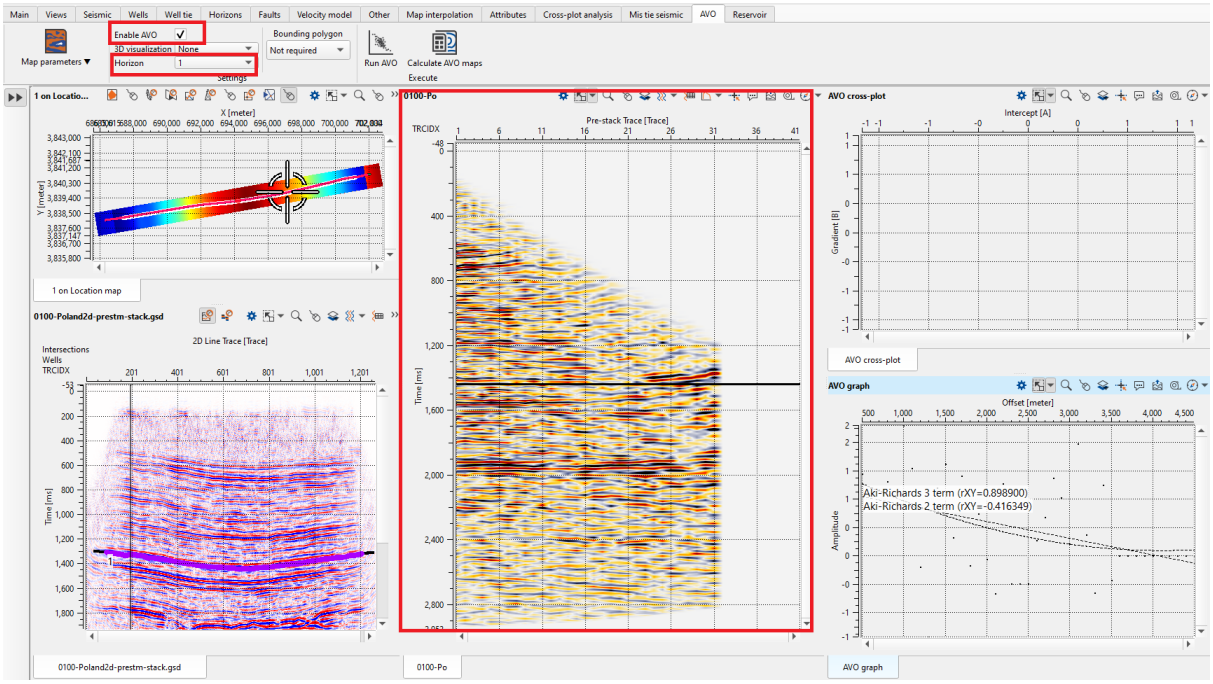
1. **Open a New Workspace**: Start a new session and add a location map from the **Views** tab.
2. **Select the Image Gather**: In the location map, click the **Selected image bin** icon to display the image bin.
3. **Display Object Items**: Go to **Views** and select the appropriate Object items such as **Image gather**, **AVO cross-plot**, and **AVO graph**.

When the **Enable AVO** option is selected, the current horizon (displayed as a black line on the image gather) will appear along with the AVO graph. The user can then run AVO calculations, which are displayed on the AVO cross-plot.

© 2026 Geomage 2003 LTD

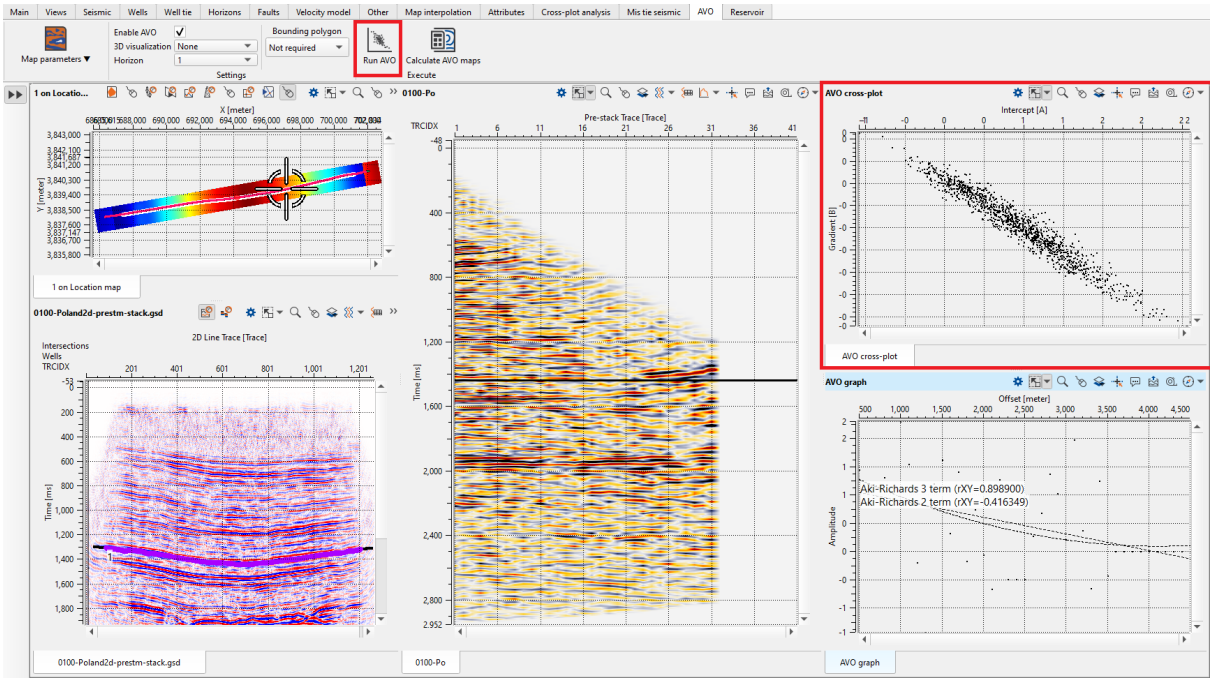


Now click on **Enable AVO** option. It will display the Current Horizon (black line on the image gather) along with AVO graph with Aki-Richards regression graphs as shown below.



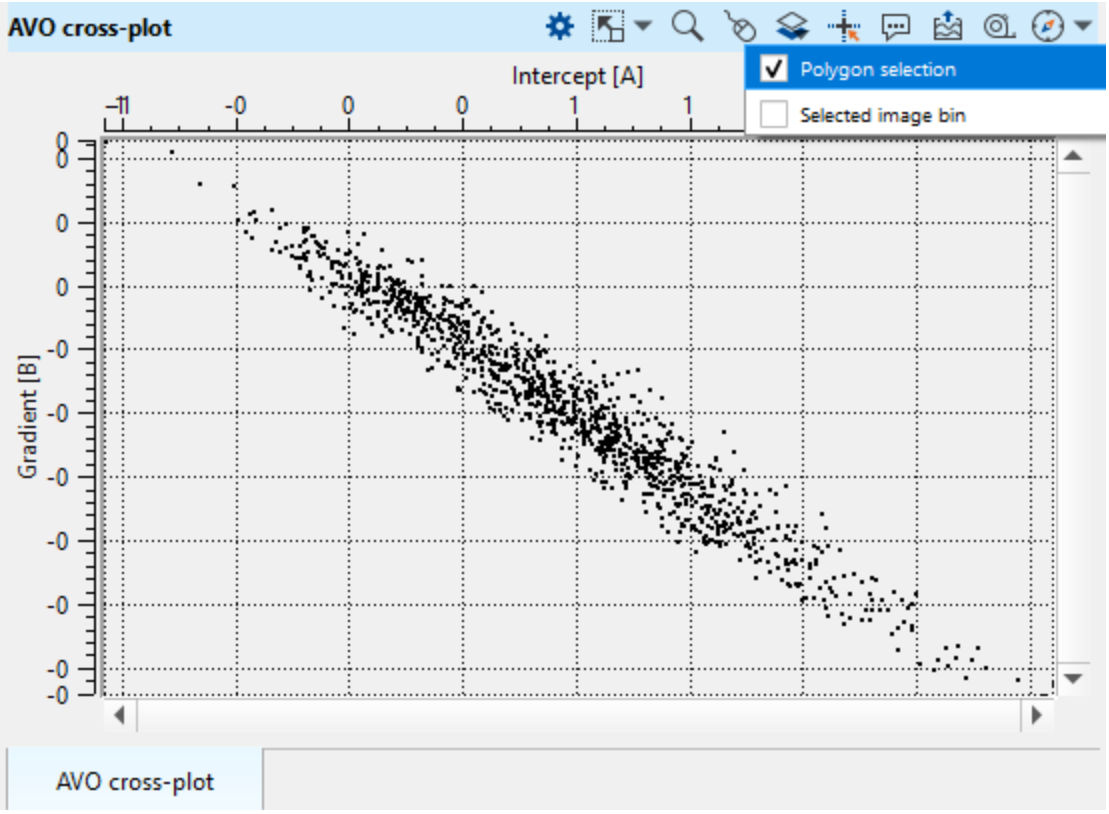
© 2026 Geomage 2003 LTD

To generate the AVO cross-plot, click on the **Run AVO** icon. It will create the cross plot as shown below.

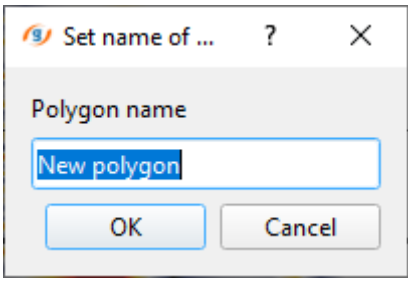


Inside the **AVO cross-plot** view, the user can pick a polygon and the corresponding bins falling inside the polygon are display on the location map. To pick a polygon, select **Control item** icon on the **AVO Cross-plot** view.

© 2026 Geomage 2003 LTD



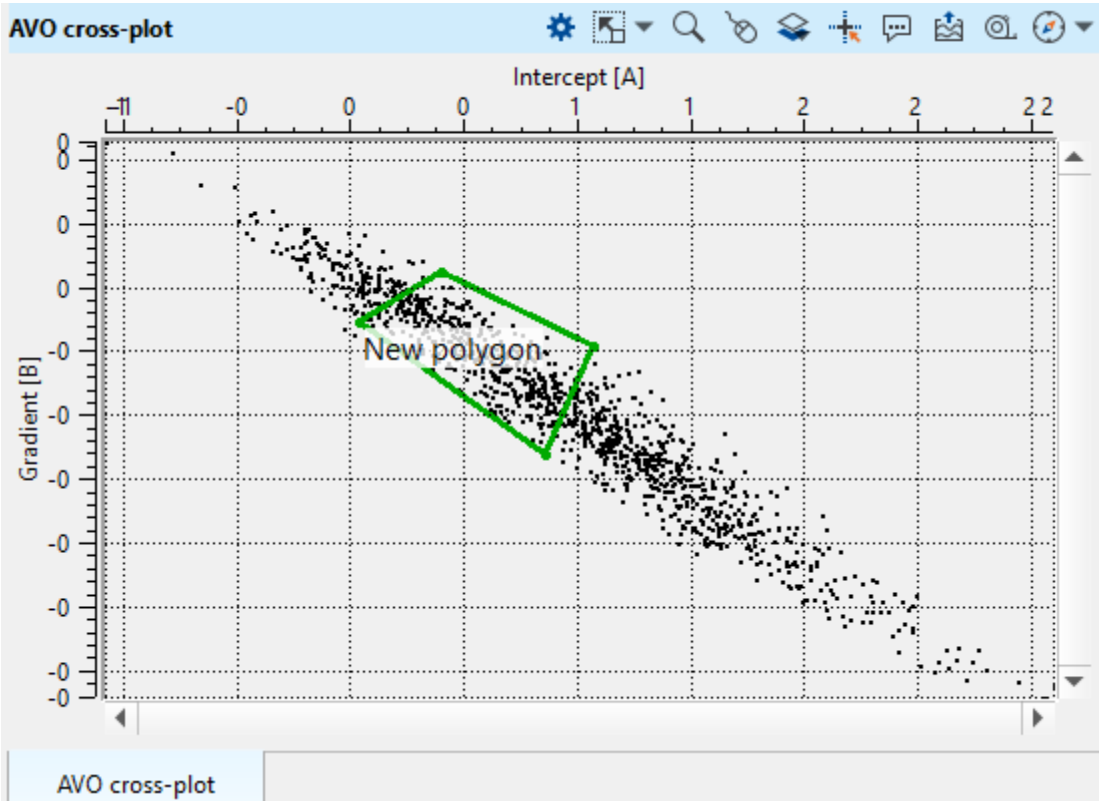
It will pop-up a window. Give a name to the polygon.



Picking a Polygon on the Cross-Plot

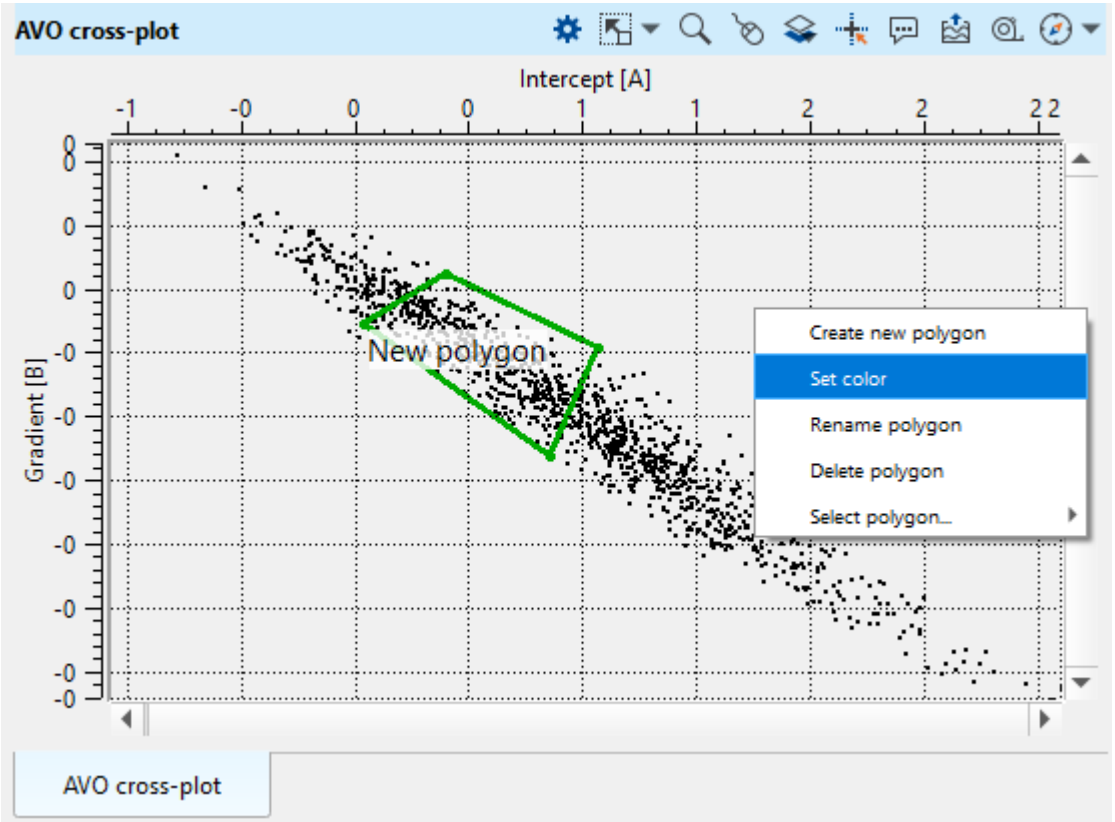
© 2026 Geomage 2003 LTD

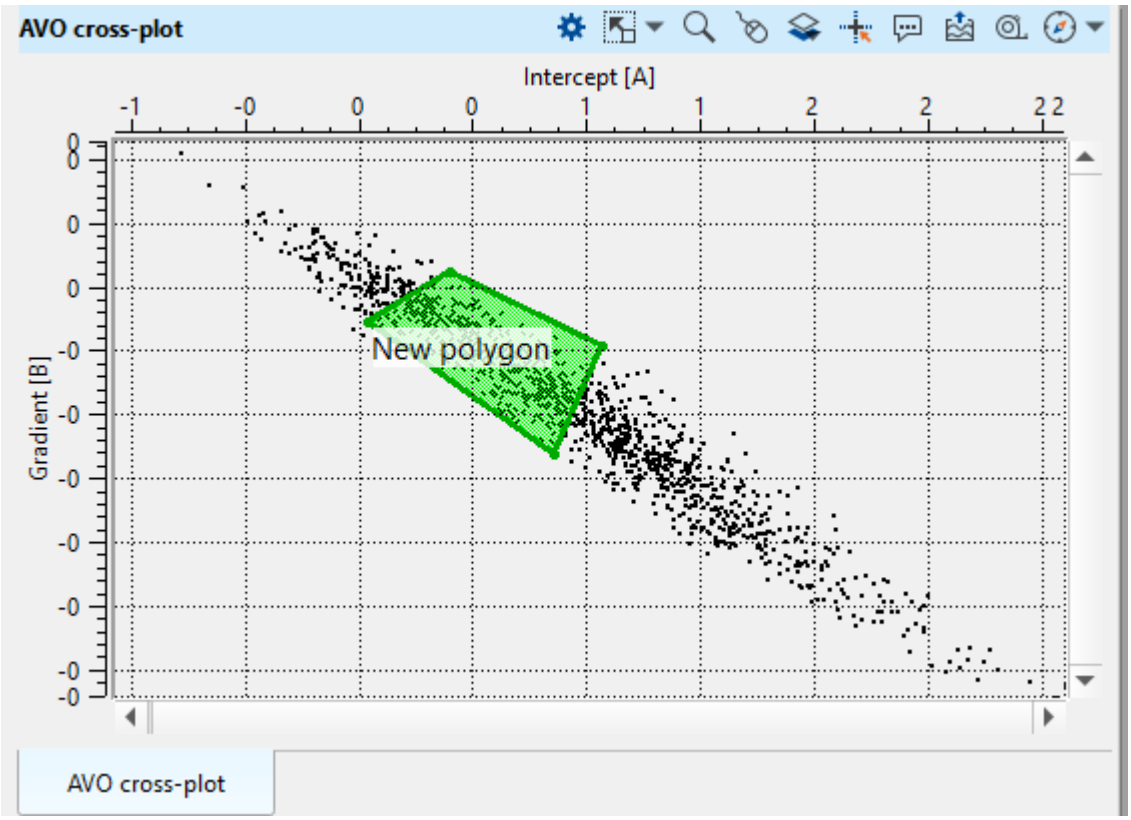
Once the AVO cross-plot is generated, users can pick polygons to analyze specific data points. Clicking on the cross-plot display allows for polygon picking, and these polygons will correspond to bins on the location map. Users can also assign different colors to polygons for better visual analysis.



Now hold **ALT+Right click** to get more options. Here the user can choose different options like create a new polygon or set a color to the existing polygon etc. We selected Set color option as shown below.

© 2026 Geomage 2003 LTD

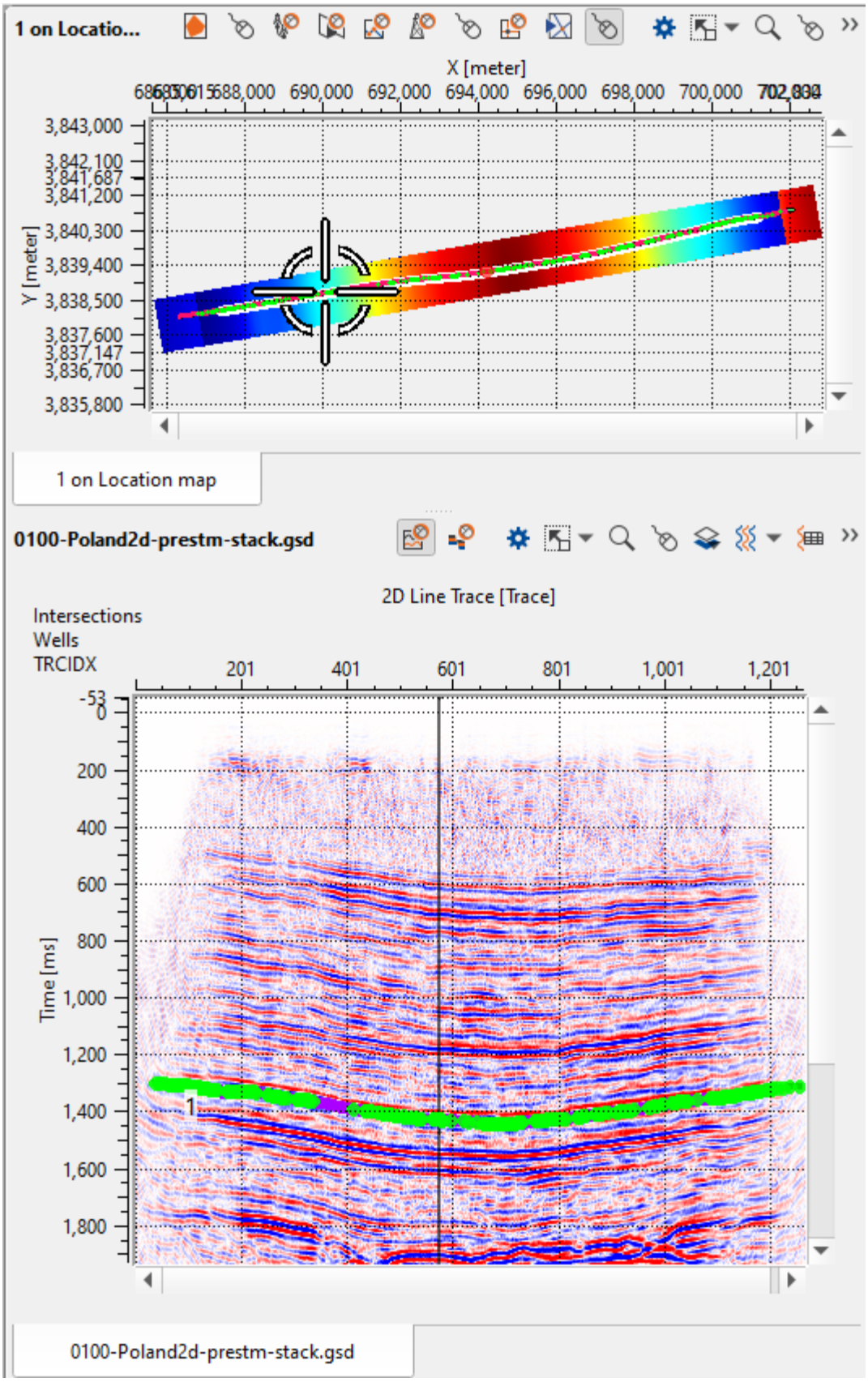




© 2026 Geomage 2003 LTD

Within the polygon, whatever the bin points present will be displayed on the location map as well as on the picked horizon. All the green dots on the location map are the corresponding bins within the polygon.

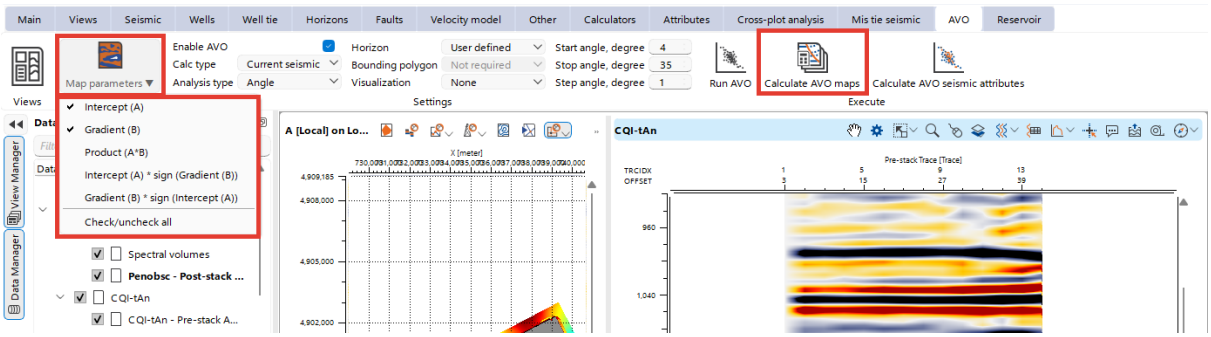
© 2026 Geomage 2003 LTD



© 2026 Geomage 2003 LTD

Calculating AVO Maps

To calculate AVO maps, such as Intercept and Gradient, the user should select the appropriate options from the **Map Parameters** drop-down menu and click **Calculate AVO Maps**. The outputs will be available under the **Visual Settings** of the location map, where the results will be stored under the **Horizon Attributes**.

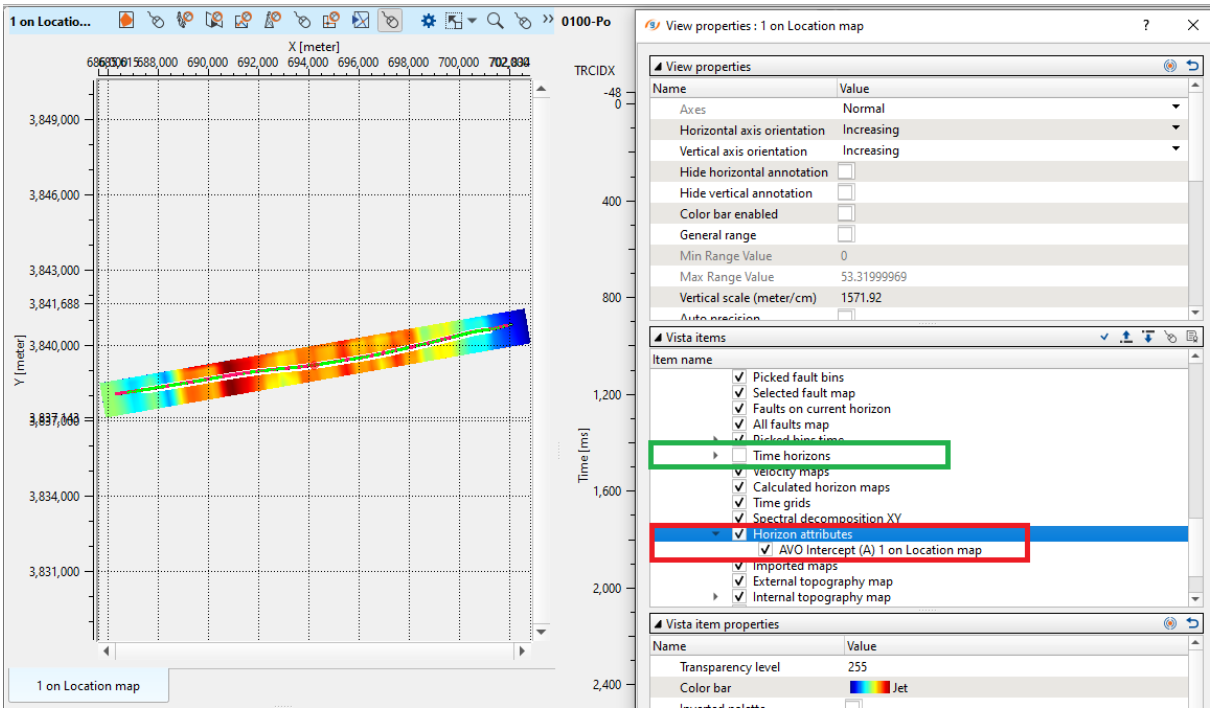


In the example below, there is the Intercept option is selected. **The output** of the **Intercept** should be available at the **Visual Settings** of the **Location map** under **Horizon Attributes**

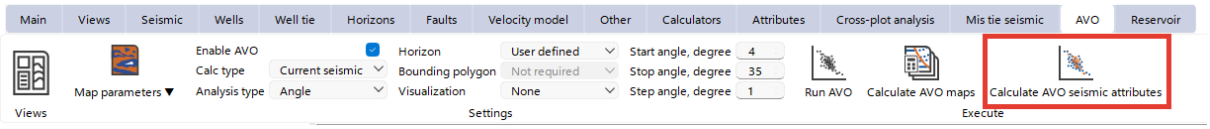
Note

To view the attributes, first uncheck the Time horizons Object items

© 2026 Geomage 2003 LTD

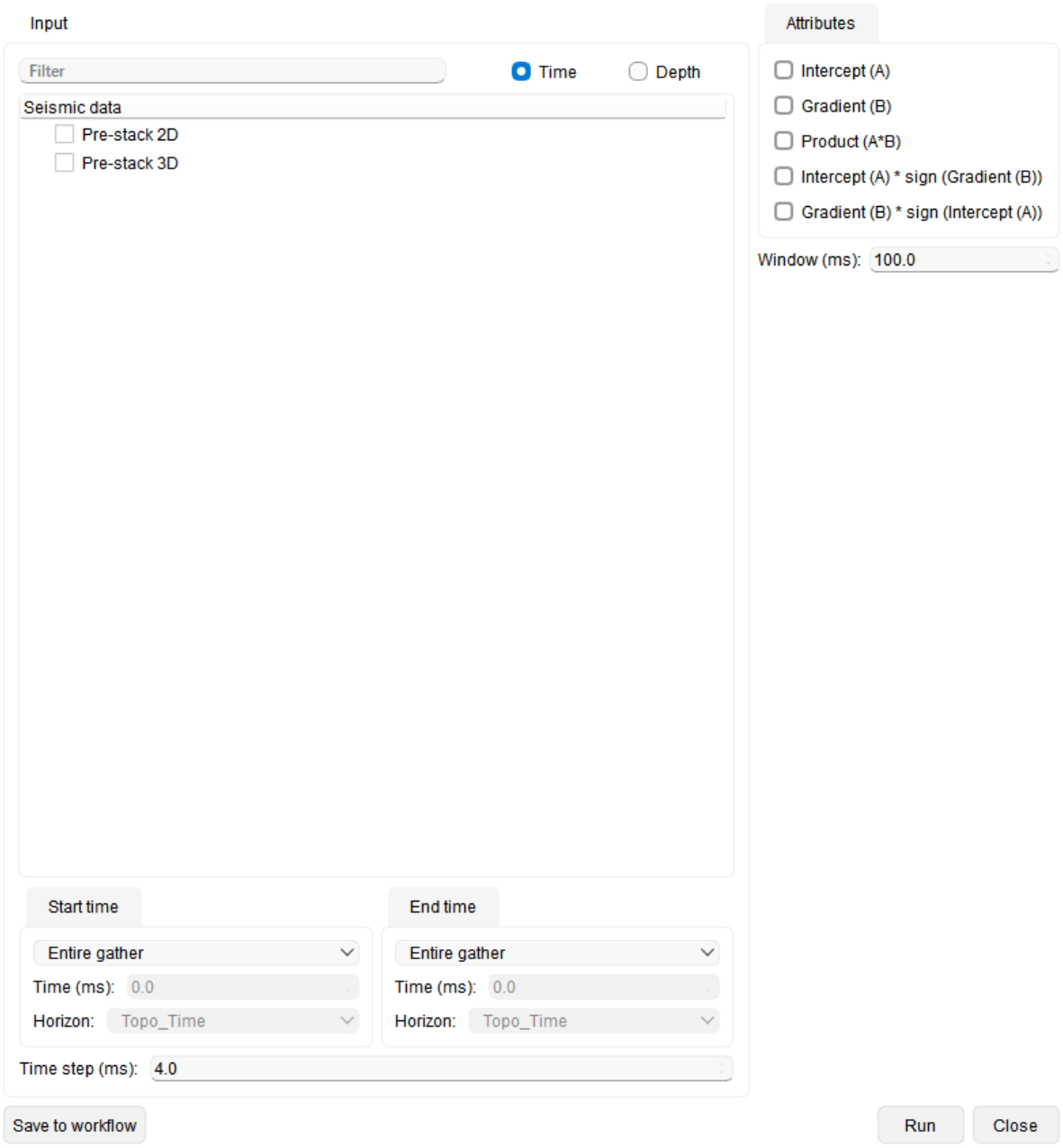


In the **Calculating AVO seismic attributes** section, users can configure and compute various AVO parameters such as *Intercept*, *Gradient*, and other derived attributes. Click on **Calculate AVO seismic attributes** in the AVO bar 119



The **Calculate AVO seismic attributes** wizard will be opened

© 2026 Geomage 2003 LTD



In the ***Seismic data*** panel, select the dataset for AVO analysis. Available data types may include *Pre-stack 2D* and *Pre-stack 3D*.

Then set the ***Start time*** and ***End time*** intervals for the AVO calculation:

Start time: Choose either a specific time in milliseconds or set a *Horizon limit* or the *Entire gather*. In the **Time** domain, the **Horizon limit** selector accepts either a time horizon or a time map.

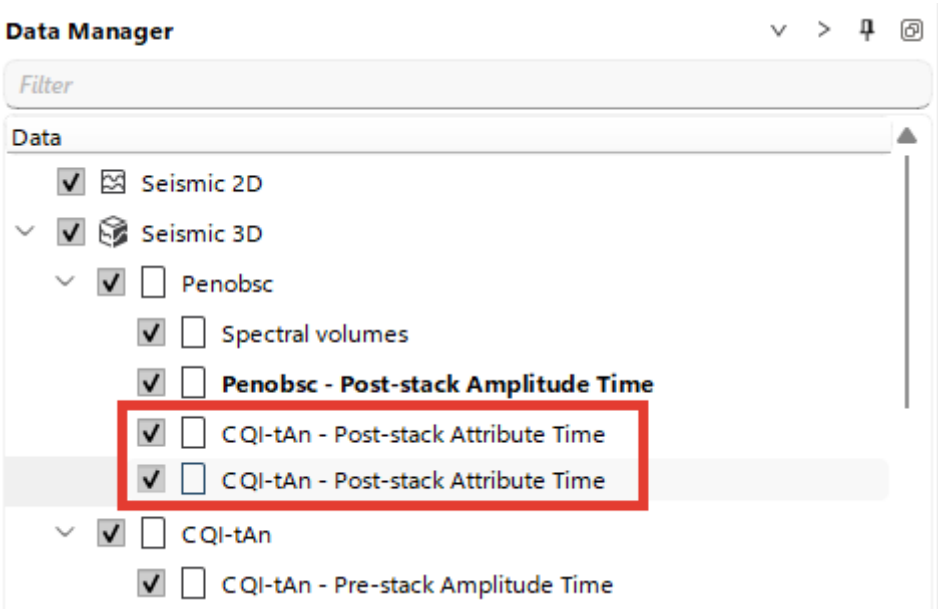
End time: Similarly, you can set the end boundary.

© 2026 Geomage 2003 LTD

In the ***Attributes* panel**, choose which seismic attributes to calculate. Define the time window (in milliseconds) over which the attributes will be calculated.

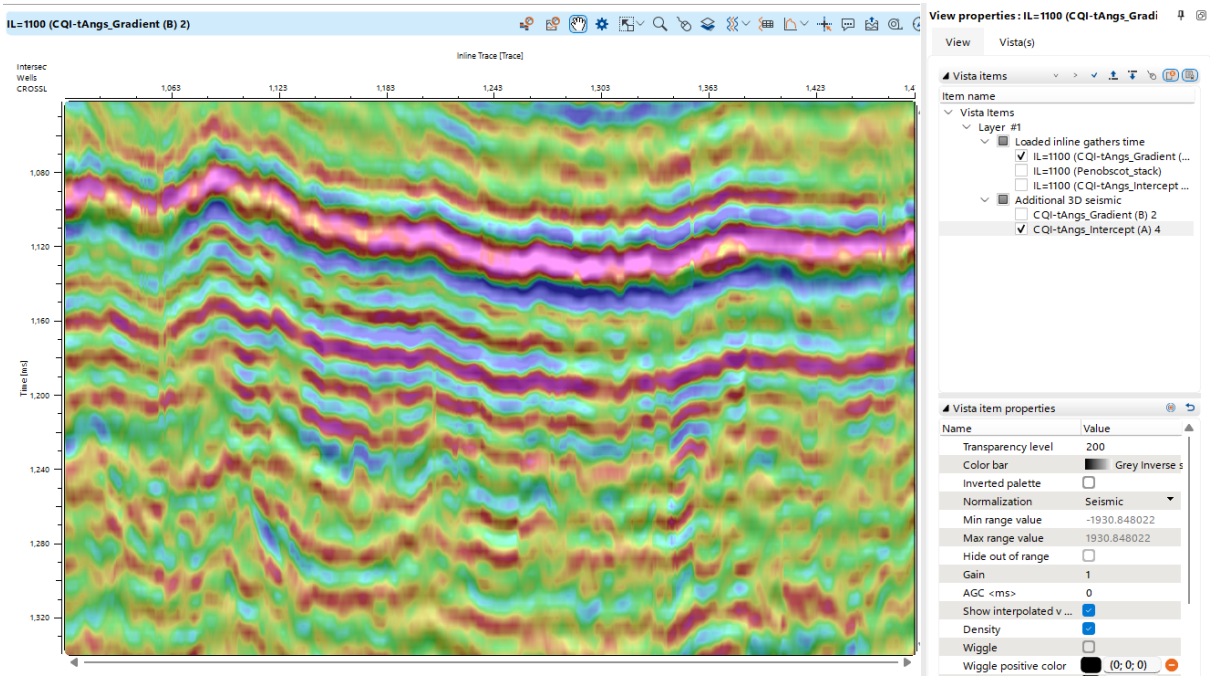
Once all parameters are configured, click ***Run*** to begin the calculation. The selected AVO attributes will be generated for the defined time interval and data range. To save the current parameters as a workflow job without running the calculation, click ***Save to workflow***. Click ***Close*** to close the wizard.

After computation, the results will be stored in the **Seismic folder** in the Data manager 124 and can be visualized in the AVO views 272.



Users can adjust visualization settings such as **transparency**, **color bar**, and **density** options in the property panel to refine the display for more detailed analysis.

© 2026 Geomage 2003 LTD

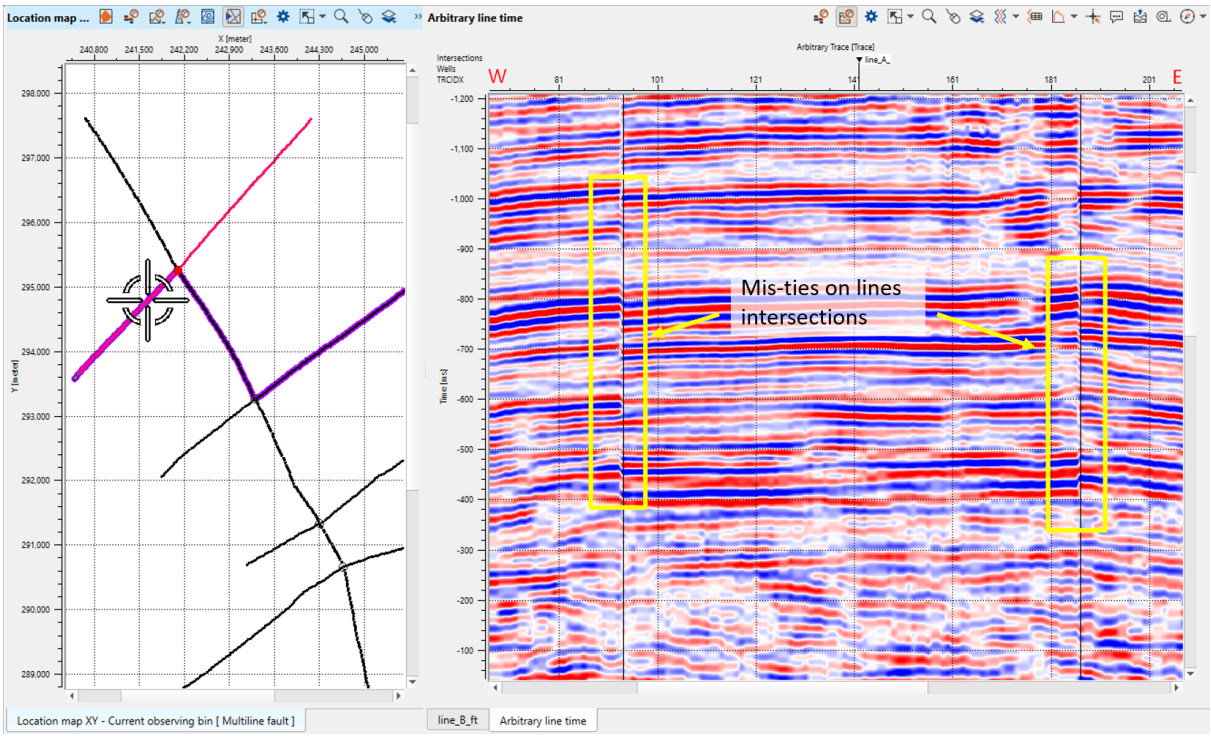


9.10 Mis-tie correction

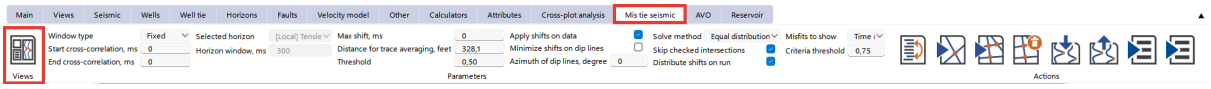
If you encounter mis-ties in your 2D seismic line set or in 3D seismic volumes, it may be necessary to apply corrections to properly correlate horizons and faults. g-Space offers powerful functionality for addressing mis-ties.

Subscribe to our channel and watch the video on how to correct mis-ties in g-Space on YouTube

© 2026 Geomage 2003 LTD



Access the Mis-tie Bar[117] and click on **Add/Show Mis-tie Views**.



This action opens a new workspace with following view types:

- **Location Map**: Visualizes the selected intersections for 2D and 3D seismic data, highlighted in yellow.

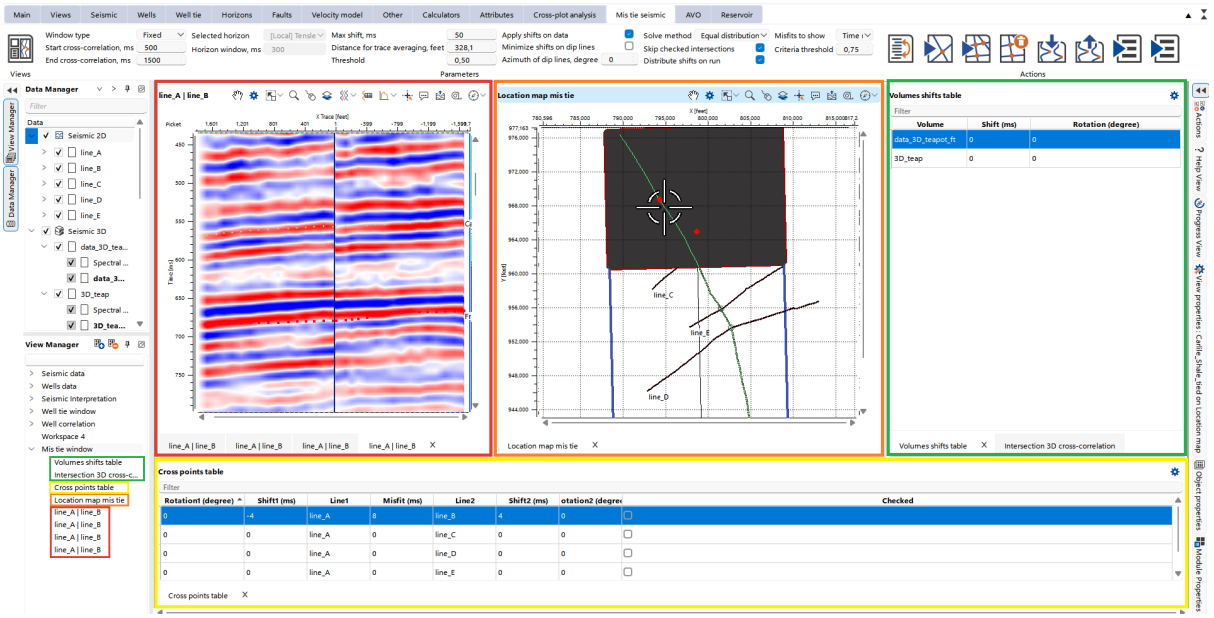
For 3D data, highlighted in green

- **Volumes shift table:** Displays all seismic volumes
- **Intersection 3D cross-correlation:** Displays correlation function between volumes

For 2D data

- **Cross Points Table:** displays all intersections with phase and time shifts, highlighted in yellow.
- **Section Views**: Provides four possible sections of the selected intersection, highlighted in red.

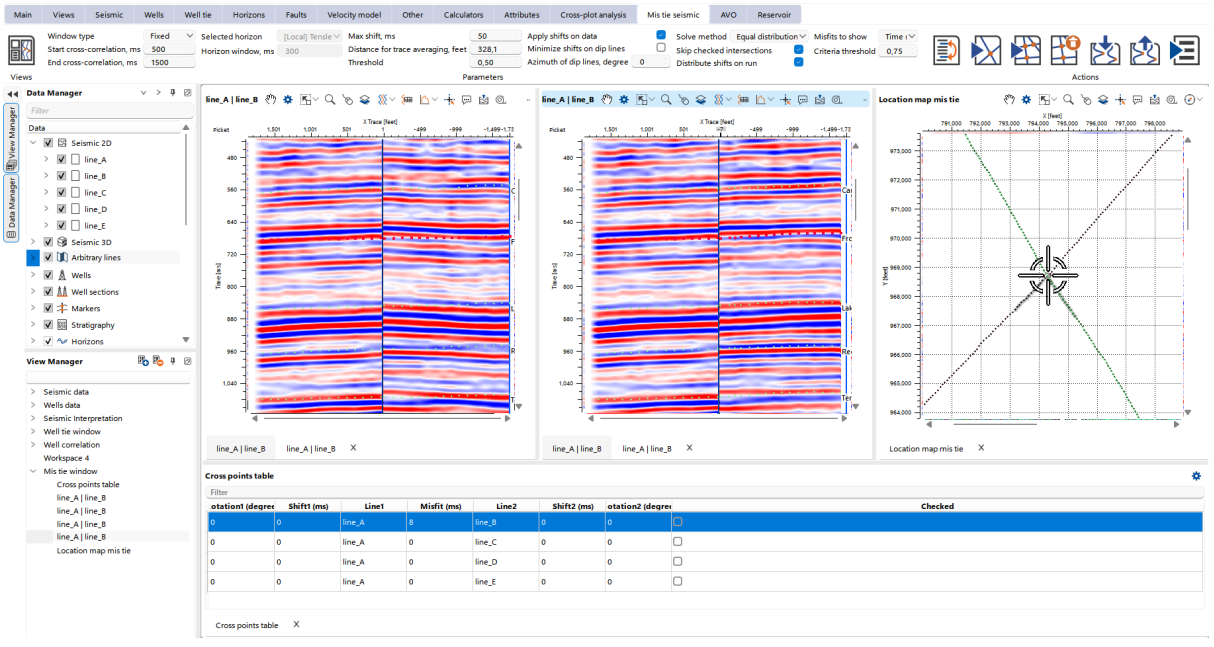
© 2026 Geomage 2003 LTD



Step-by-step workflows can be found further in the chapters for 2D lines 715 and 3D volumes 720.

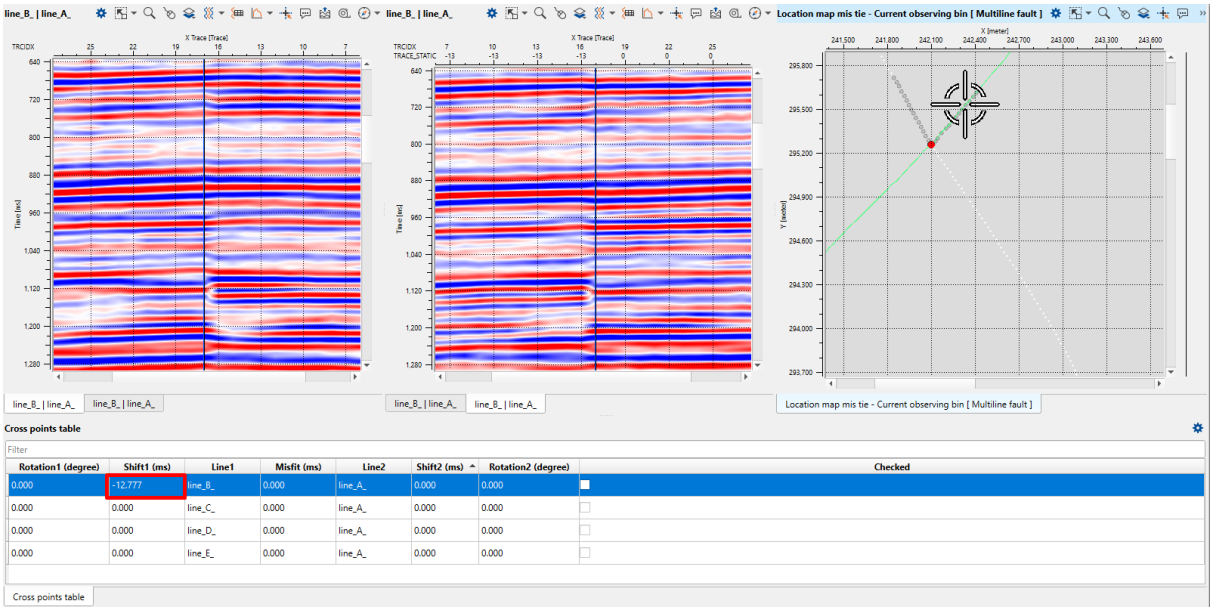
9.10.1 2D mis-ties correction

Arrange your workspace as desired, ensuring relevant views (e.g., seismic sections, location maps) are visible.

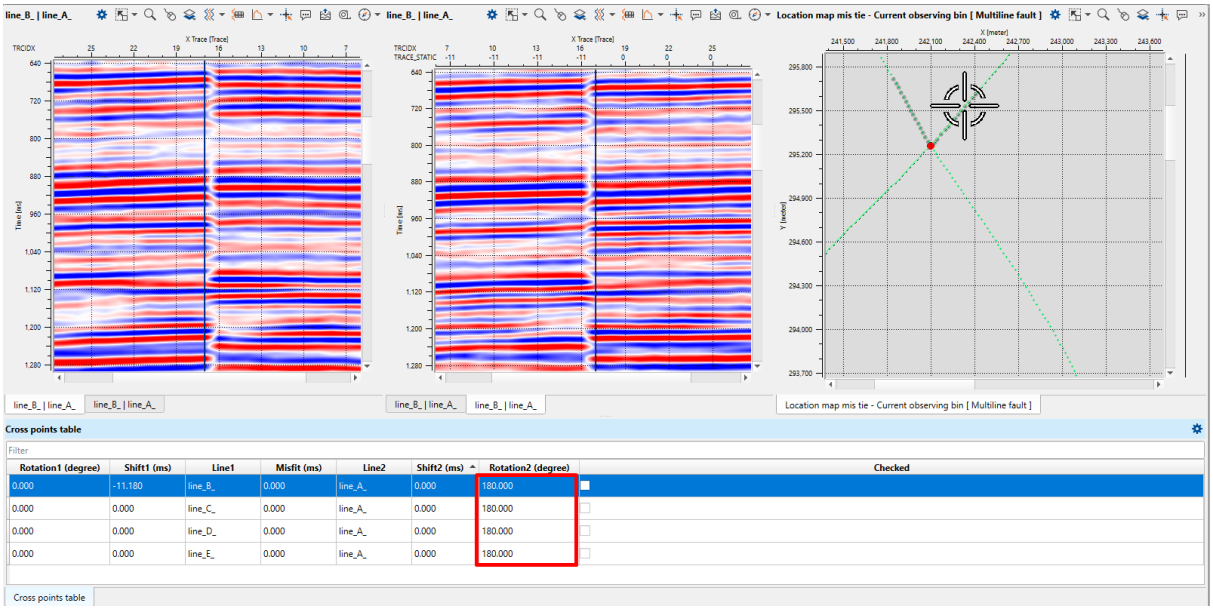


© 2026 Geomage 2003 LTD

Use the **left mouse button** to drag the seismic section up or down, applying shifts to specific parts of the section. These shifts will be recorded in the **Cross Points Table**.

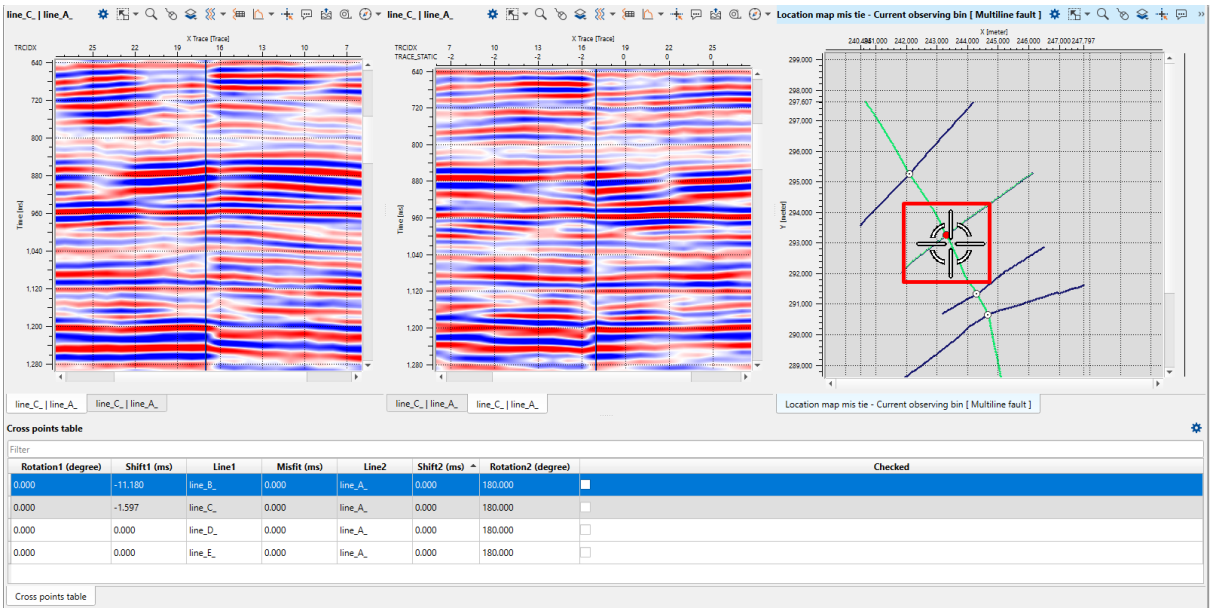


To clear shift click by right mouse button on the section.
To add phase rotation put the value to the cross point table

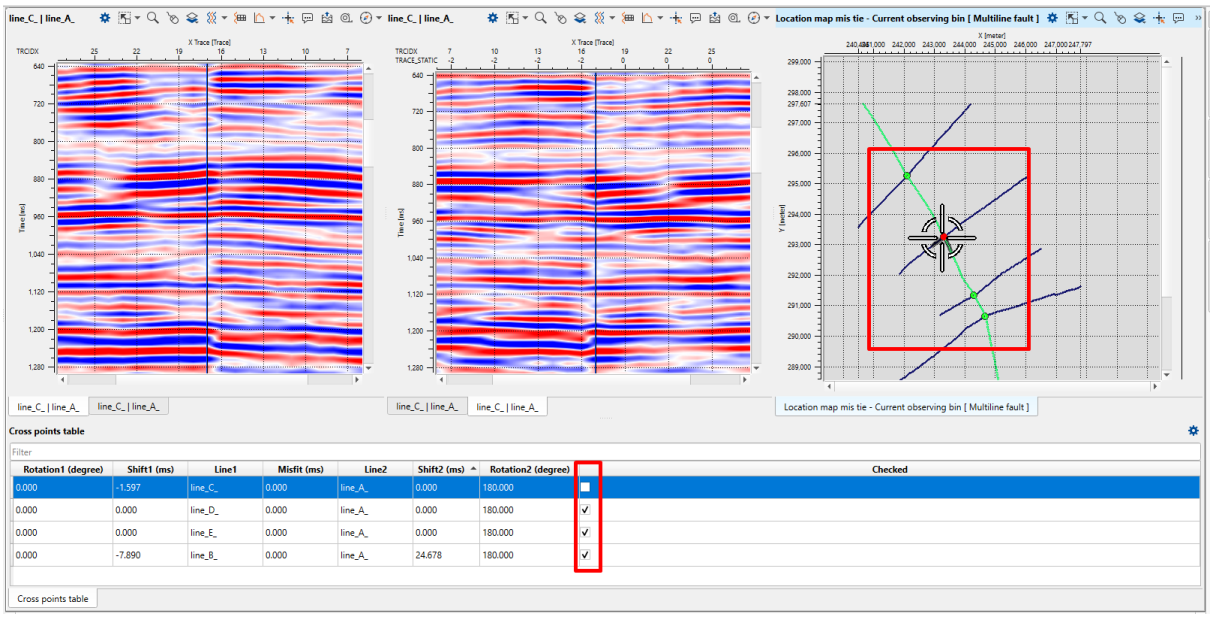


Once adjustments are complete, move to the next intersection via the location map or by selecting the corresponding entry in the **Cross Points Table**.

© 2026 Geomage 2003 LTD



After finishing work on an intersection, you can mark it as "checked" by double-clicking the relevant box. Checked intersections will be highlighted in green on the map.

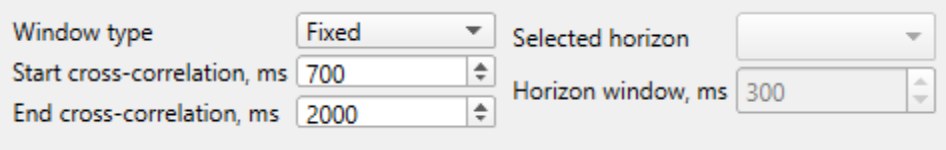


Automatic Mis-tie Correction: In addition to manual corrections, automatic mis-tie correction through time-shift calculation is available. This method leverages cross-correlation and requires specific settings input in the **Mis-tie Seismic** toolbar:

1) **Cross-correlation settings:**

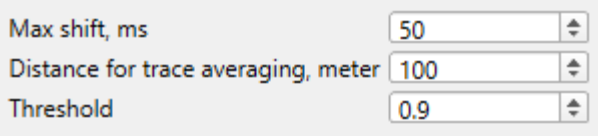
© 2026 Geomage 2003 LTD

- Choose between a fixed time window or a window centered around a particular horizon. If a horizon is selected, it will form the center of the specified window. The selected horizon must already have a map; otherwise the calculation stops and asks you to create a map for that horizon first.



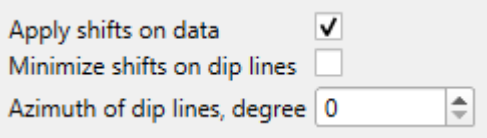
2) **Time-shift parameters:**

- Set the maximum allowable time shift.
- Specify the distance for trace averaging, which will be used for comparison.
- Define the threshold for adding traces from the specified distance to the average.



3) **Applying shifts:**

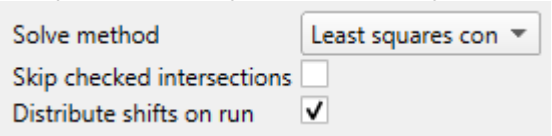
- Enable shifts to be applied to the seismic data.
- Optionally, activate shift minimization for lines with a specific azimuth, useful in cases involving dipping horizons where azimuth alignment can improve results.



4) **Shift calculation methods:** Select from various methods:

- **Equal distribution:** Splits mis-tie values evenly between intersecting lines.
- **Least squares:** Minimizes the sum of squared mis-tie corrections.
- **Least squares with constant time:** Same as above but with a constraint that the time shift is constant for each line.
- **Least squares with constant time + phase:** Includes phase shifts in the calculation.
- **Polarity estimation:** Estimates and corrects polarity differences between intersecting lines.

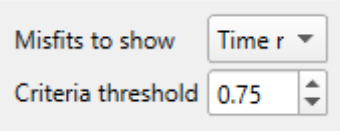
Optionally, skip intersections marked as "checked" in the calculations. Also, choose whether to apply shifts directly to the data or just visualize them on the map.



5) **Mis-tie visualization:**

© 2026 Geomage 2003 LTD

- Choose whether to visualize mis-ties on the map based on **time** or **phase**.
- Set a **threshold** to halt the binding process once a certain accuracy is achieved.



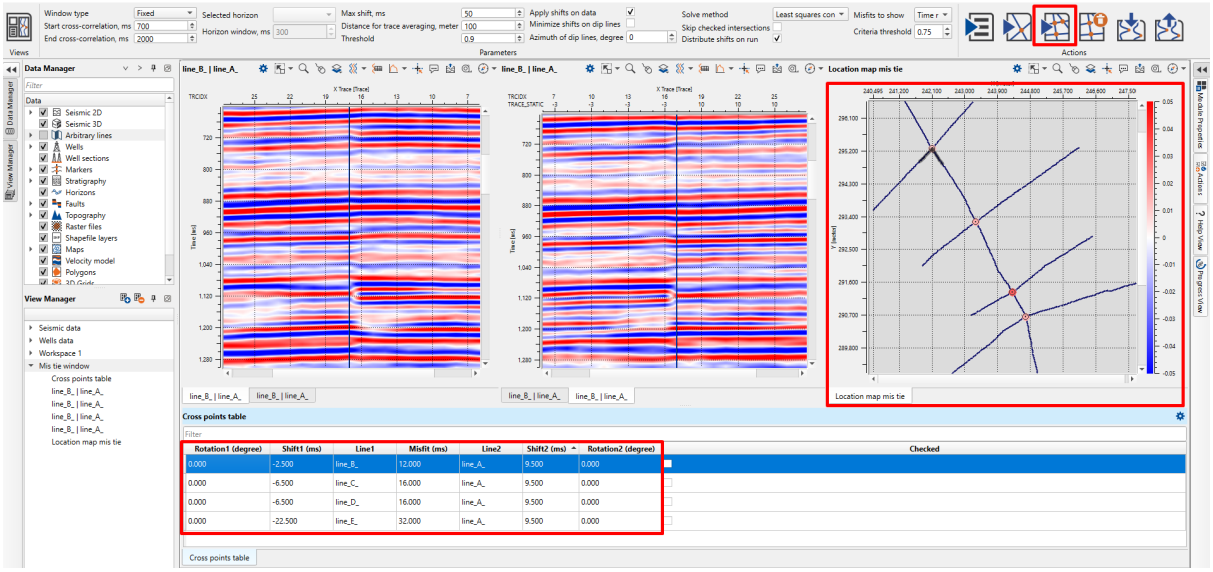
6) **Group priority:**

- If multiple seismic groups exist, prioritize them for mis-tie correction using the appropriate button.

You can perform shift calculations on individual intersections or across the entire dataset:

When calculating across all lines, the **Cross Points Table** will automatically populate with the calculated shift values.

Enable **Misfit Points** in the properties menu, which will be color-coded based on their mis-tie value, and displayed on the location map.



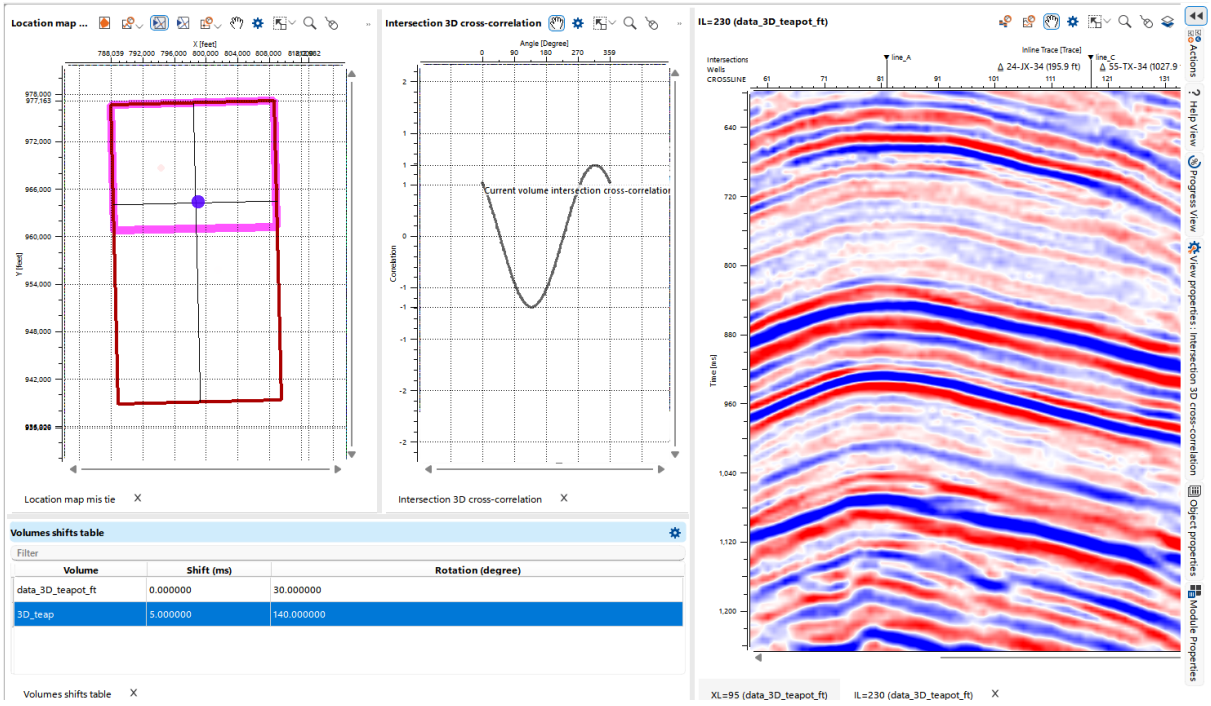
© 2026 Geomage 2003 LTD

9.10.2 3D mis-ties correction

The **3D Mis-ties Correction** tool in g-Space enables the adjustment of seismic datasets for time and phase shifts between intersecting 3D seismic volumes. This correction improves the consistency between datasets, particularly when dealing with different acquisition or processing workflows.

To begin, configure your workspace as demonstrated in the image below, ensuring that the **Location map**, the **Volumes shifts table** and the **Intersection 3D cross-correlation** views are visible.

In the image below, the Location map shows two volumes in different colors. It is also convenient to view the results interactively in the section view.

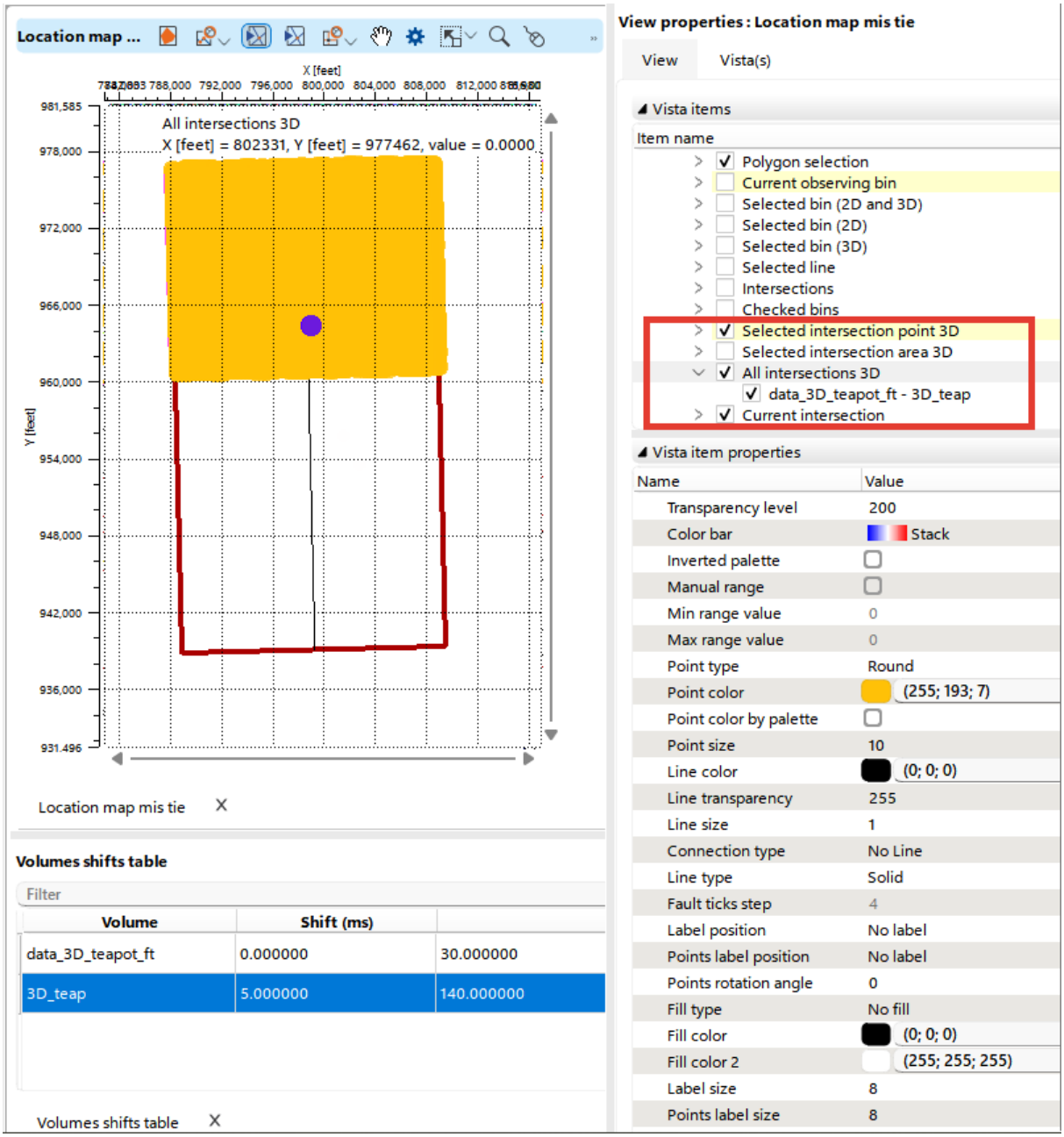


If your project includes multiple seismic processing groups, the priority for applying corrections can be adjusted using the **Seismic Group Priority** button

Calculate the intersections of volumes by clicking the **Calculate volumes intersection** button

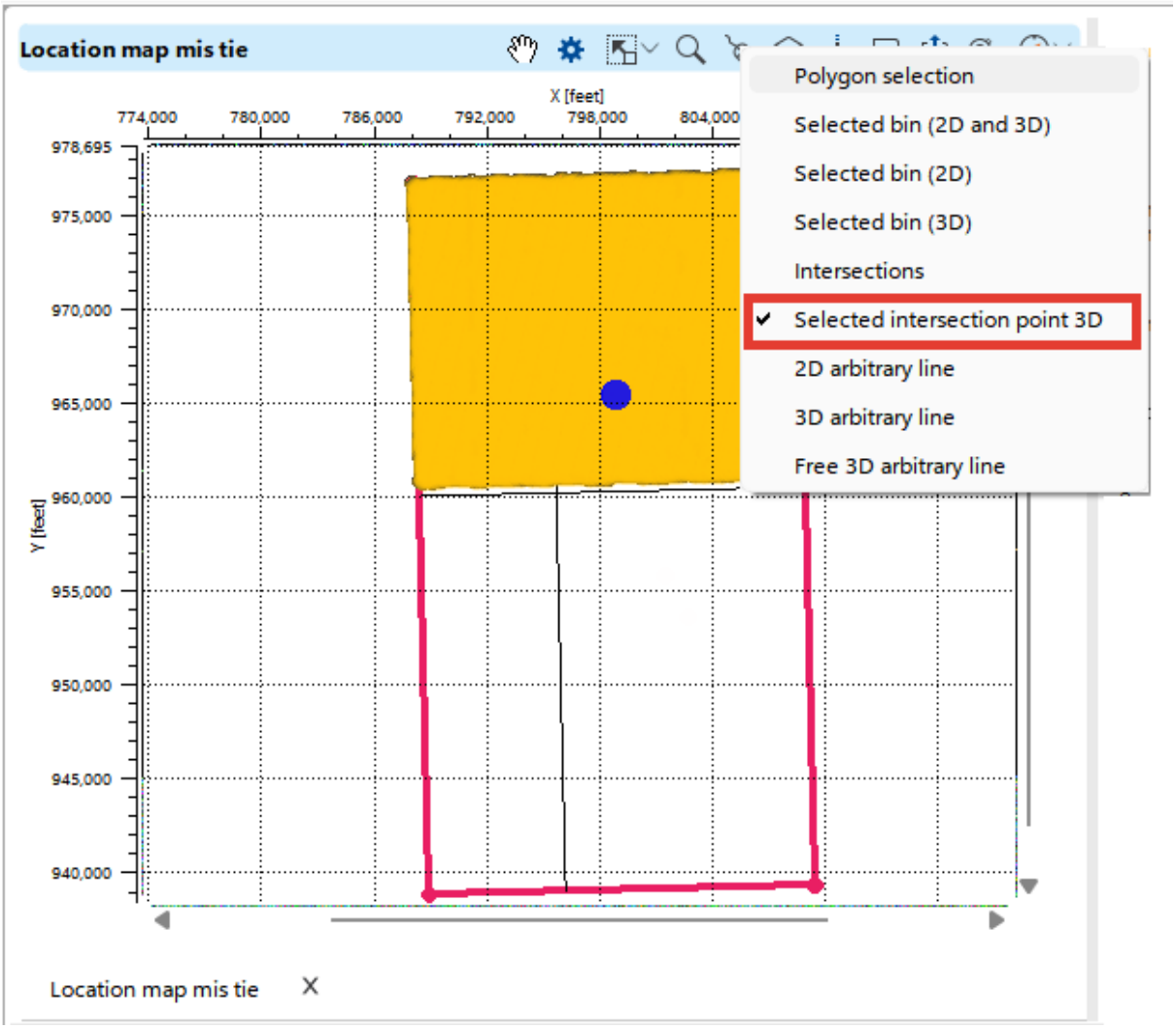
in the bar, new objects will appear in **Object(s)**

© 2026 Geomage 2003 LTD

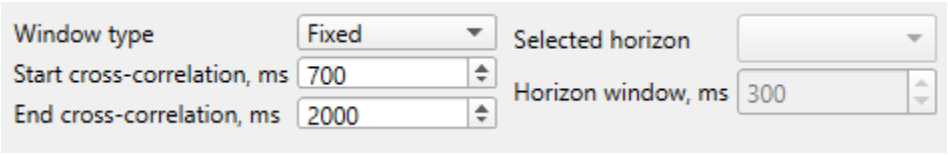


Select the **intersection point 3D,** at this point the mis-ties and cross-correlation will be calculated.

© 2026 Geomage 2003 LTD

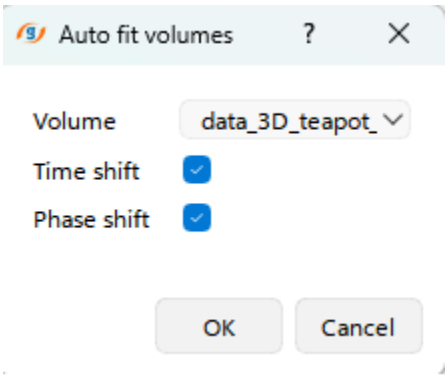


Choose between a fixed time window or a window centered around a particular horizon. If a horizon is selected, it will form the center of the specified window. The selected horizon must already have a map; otherwise the calculation stops and asks you to create a map for that horizon first.

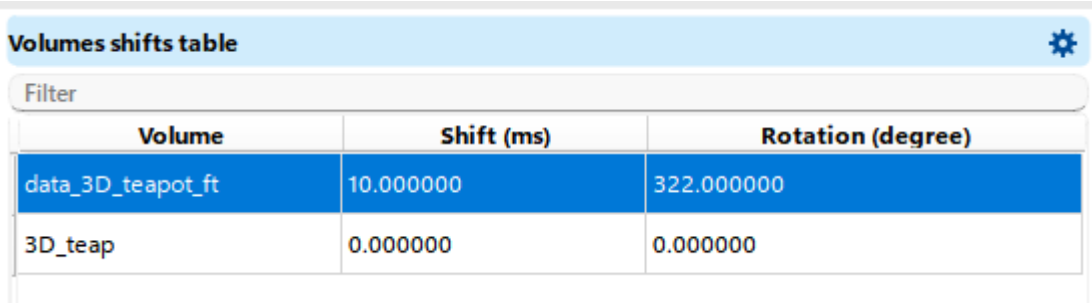


Then click **Auto fit volume** button in the bar and in the pop-up window that appears, select the volume relative to which the correction and the time, phase, or both variants of the residuals will be calculated.

© 2026 Geomage 2003 LTD

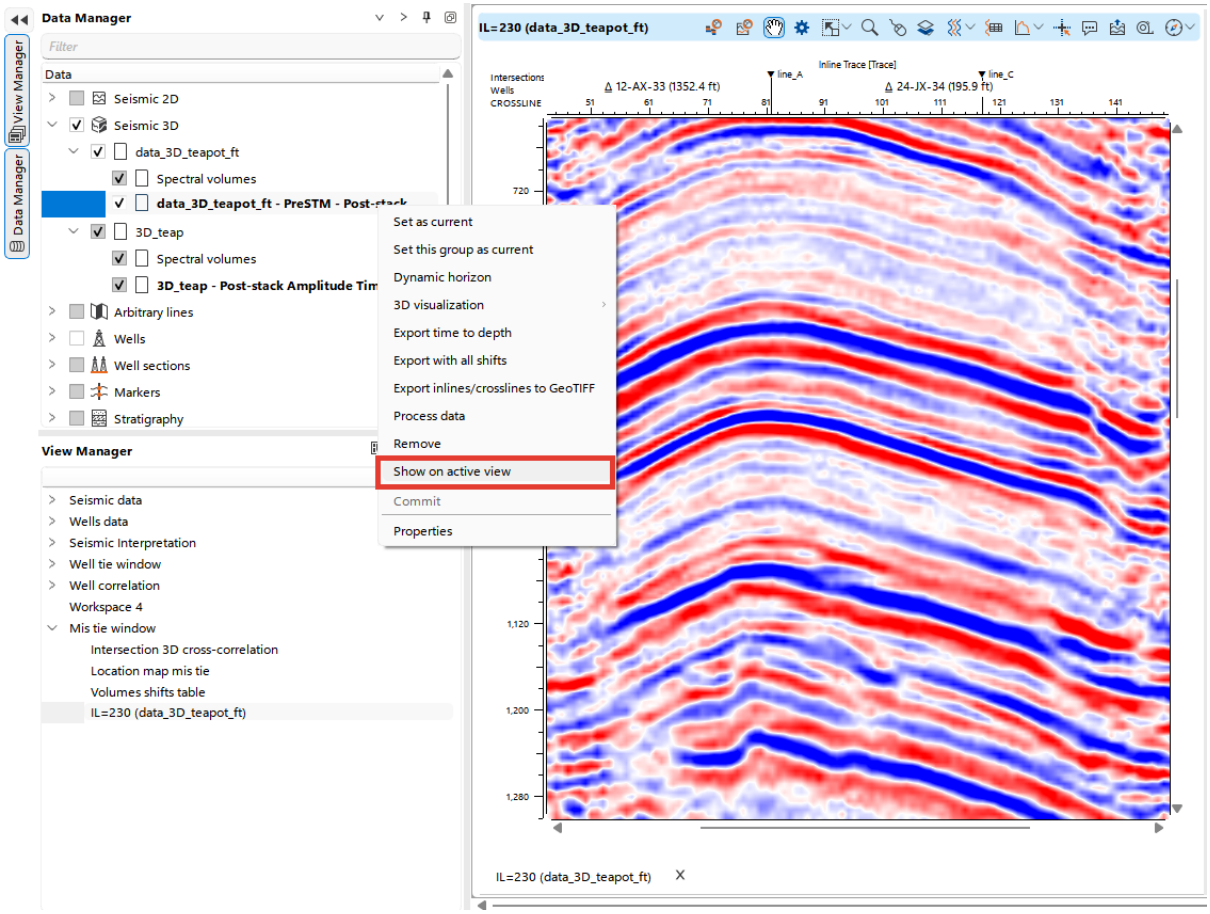


The calculated shifts will appear in the Volumes shift table.
If necessary, you can enter the time and phase shifts into the table manually.
Once the calculation is done, the shifts will be applied to the seismic volume.



To efficiently visualize the results, you can use the **Show in Active Window** function available in the Data Manager for seismic data. This option allows seamless display of the selected seismic data within the active visualization window. For more detailed information on how to display and work with seismic data, refer to the Seismic Data Visualization 587 chapter in this manual.

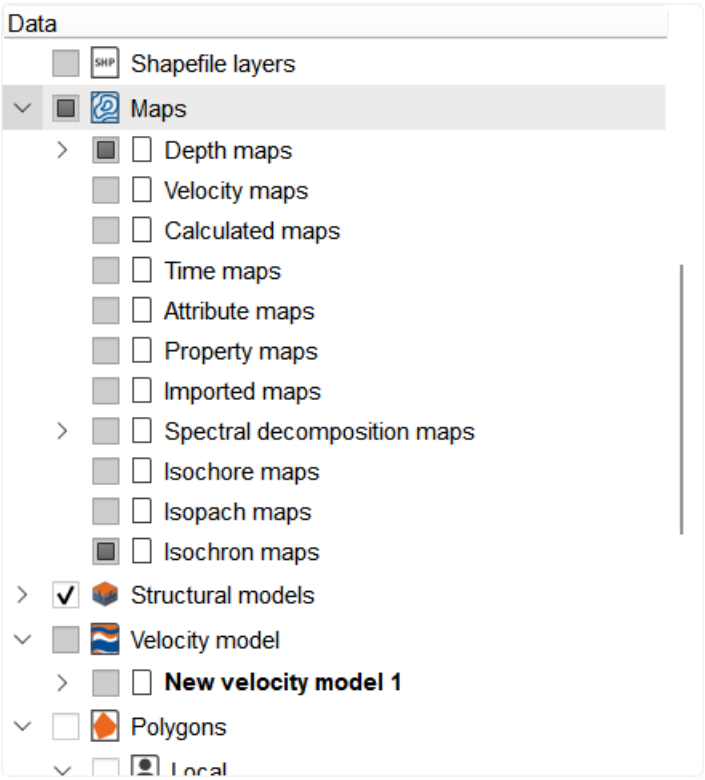
© 2026 Geomage 2003 LTD



9.11 Mapping

g-Space provides a variety of preinstalled map folders within the **Data Manager** and allows users to modify settings in **Module Properties** for customized outputs.

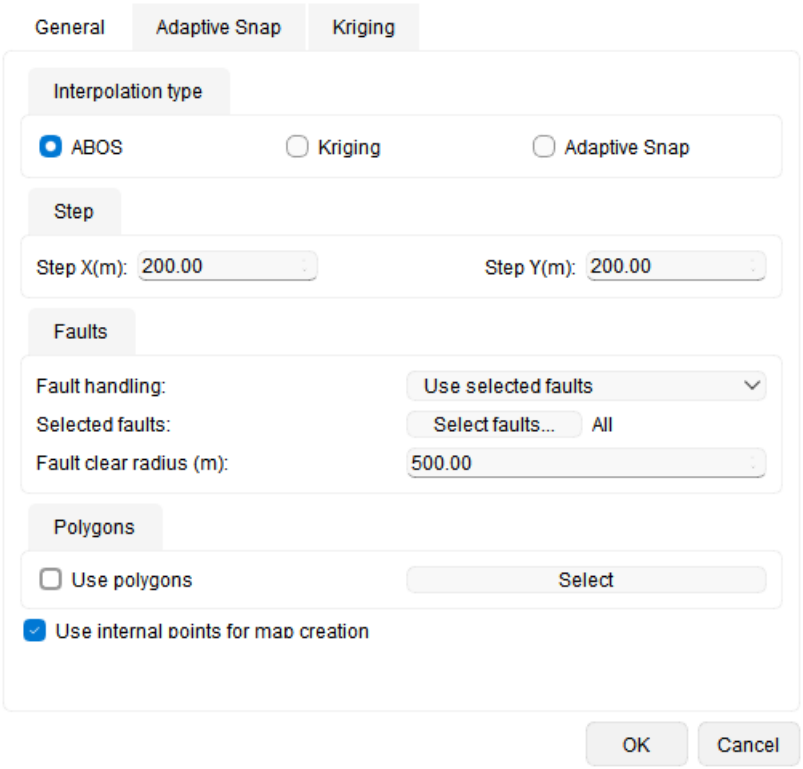
© 2026 Geomage 2003 LTD



Interpolation settings for creating maps are available on the Modelling ribbon tab, in the **Map calculator** group. Click **Settings** to open them.

On the **General** tab, set the interpolation type, grid steps, fault handling, polygons, and internal-point options.

© 2026 Geomage 2003 LTD



In **Kriging** tab you can set covariance, range and max points

Advanced interpolation settings are available in the Module Properties 197.

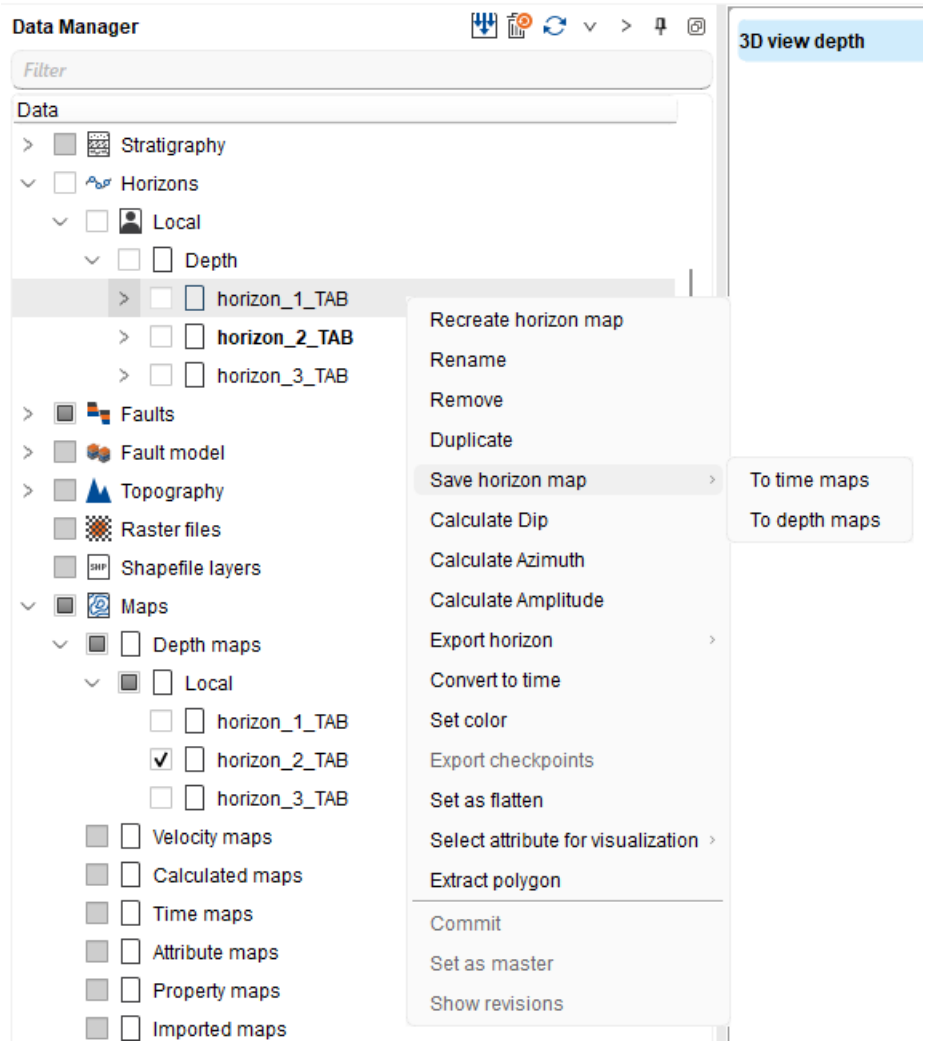
The g-Space map creation process is highly customizable, with options to adjust settings for different geological scenarios. By using preinstalled map folders, applying appropriate interpolation methods, and incorporating faults and polygons, users can generate detailed and geologically accurate maps.

Mapping in g-Space allows you to generate and edit maps of various geological and geophysical properties. This process includes creating maps 727 from horizons, applying velocity models to generate depth and thickness maps etc., and editing maps 733 through the Map Editor tools.

© 2026 Geomage 2003 LTD

9.11.1 Map creating

The **Horizons** folder in the **Data Manager** stores data as Points and Maps. When a horizon is picked, only points are created initially. To generate a map, right-click on the horizon and choose **Recreate horizon map**. You can then save the generated map to the Maps folder by selecting **Save horizon map** from the Horizons folder and choosing either the time or depth scale, depending on the type of data you are working with. To convert time horizons to depth maps, a velocity model must be present in the project. By default, the active velocity model is used for this conversion.



The user can set the mapping settings in the **Modelling** ribbon tab, in the **Map calculator** group, by clicking the **Settings** button, as well as in the Module Properties. The interpolation parameters are described in Modelling 100.

© 2026 Geomage 2003 LTD

To convert time maps into depth maps and back, a Velocity model 658 must be created in the project. By default, the active Velocity model is used for conversion.

A map can also be built from a horizon, from well markers or from a point set with the Map building 730 dialog, which keeps its own set of parameters and saves the result in the Calculated maps folder.

Thickness maps

Thickness maps are stored in the **Maps** folder of the **Data Manager**, in a separate folder for each thickness type:

Isochron maps — time thickness between two surfaces in the time domain.
Isochore maps — vertical thickness between two surfaces in the depth domain.
Isopach maps — true stratigraphic thickness, measured perpendicular to the bedding.

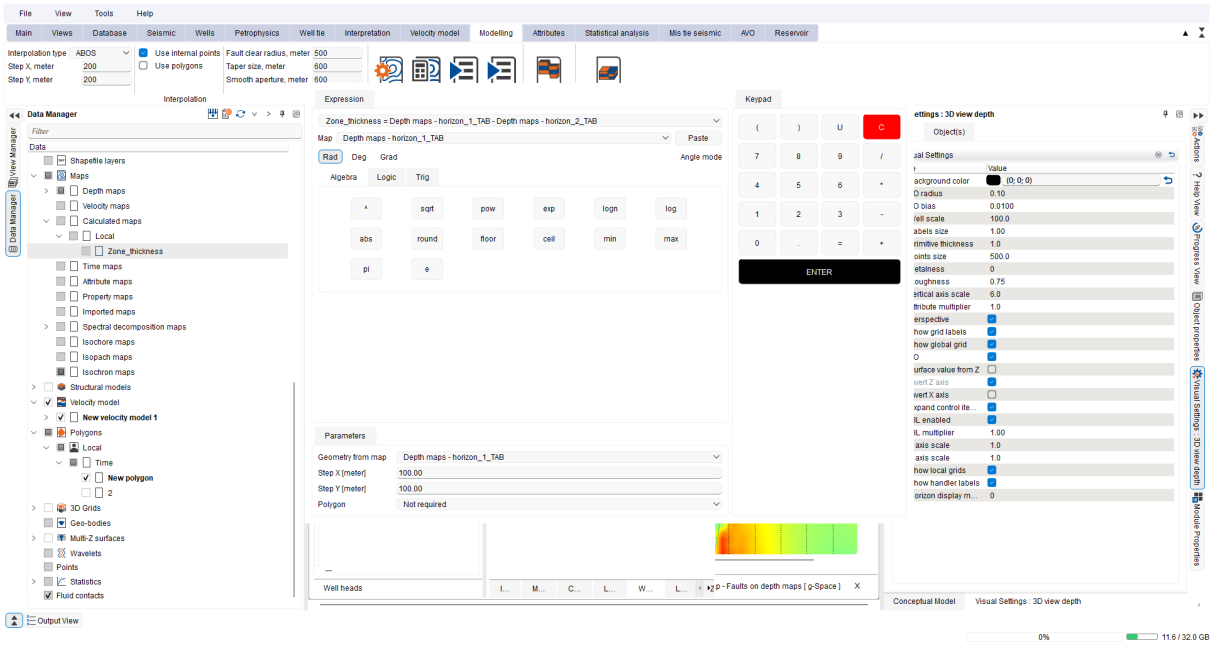
To build thickness maps from picked horizons, right-click the **Time** or **Depth** node in the **Horizons** branch of the **Data Manager** and choose **Create thickness maps**. The command is available when the domain holds at least two horizons, and each pair of adjacent horizons produces one map. Time horizons produce **Isochron maps** and depth horizons produce **Isochore maps**; the map name starts with **Isochron**_ or **Isochore**_ and is completed with the names of the two horizons.

For depth horizons the same menu also offers **Create isopach maps**, which builds the true stratigraphic thickness for each pair of adjacent horizons and saves it in **Isopach maps** under a name starting with **Isopach**_.

Maps extracted from a 3D grid zone are stored the same way. In the **Extract map** dialog, the **Map type** list offers **Isochore map** for a depth grid and **Isochron map** for a time grid, and a depth grid additionally offers **Isopach map**. The read-only **Storage location** field shows the folder the result will be saved in, and the map name is built from the grid and zone names, for example **Depth_grid_Frontier_Isochore**.

The **Map Calculator** allows users to create new maps by performing calculations using existing map data. To open the **Map Calculator**, navigate to the **Modelling** ribbon tab, **Map calculator** group, and click

© 2026 Geomage 2003 LTD



In the **Map** list, select an input map and click **Paste** to add it to the expression. Every entry in the list is named after the map type folder that holds the map, followed by the map name. Repeat this for every map the expression uses.

In the **Expression** field, define the mathematical operation or formula you want to apply to the selected maps. Type the name of the result map at the start of the expression, followed by an equals sign, for example **Zone_thickness =**. Without a result name the calculation is not started.

Refer to the Trace Headers Math 521 for a description of the available mathematical functions.

Use the **Geometry from map** list to choose the map whose grid geometry the result adopts. It must be set whenever the expression uses more than one map; with a single map in the expression the geometry of that map is used.

Define the grid step sizes in the **Step X** and **Step Y** fields. They set the resolution of the resulting map and are ignored when **Geometry from map** is set.

If you wish to restrict the calculation to a specific area, select a bounding polygon. If no restriction is needed, leave this as "Not required."

The history of the expressions used in the calculation is displayed for reference, allowing users to track their operations.

Once the expression and the parameters are configured, click **ENTER** in the keypad to generate the new map. The resulting map will be saved in the **Calculated Maps** folder in

© 2026 Geomage 2003 LTD

the Data Manager. In the data manager you can drag and drop maps from Calculated Maps to any other folders

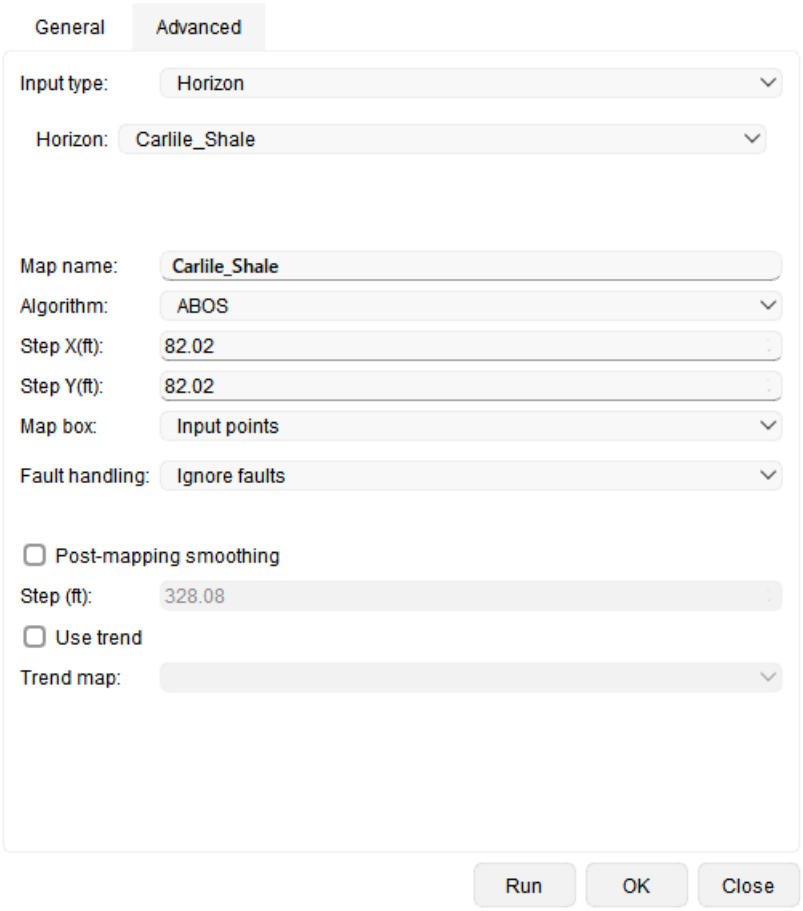
In g-Space, users can calculate and visualize attribute maps. A detailed workflow is described in the Seismic Attributes 680 chapter.

9.11.2 Map building

Map building builds a map from a horizon, from well markers or from a point set in a single dialog, with its own set of parameters. Open it on the **Modelling** ribbon tab, in the **Mapping** group, by clicking **Map building**.

The dialog stays open while you work, so you can build a map, look at the result and build the next one without reopening it. The parameters are kept with the project. The **General** tab holds the input data and the main build parameters, and the **Advanced** tab holds the settings of the selected algorithm.

© 2026 Geomage 2003 LTD



Input data

Input type sets the kind of data the map is built from. The controls beneath it change with the selection:

- **Horizon** — select one of the project horizons in the **Horizon** list.
- **Well markers** — select a marker of the active marker set in the **Marker** list, then the value to map in **Attribute**. The attribute list offers **MD**, **TVD**, **TVDSS** and the attributes of the marker itself; **TVDSS** is used by default.
- **Points** — select a point set in **Point set** and one of its numeric columns in **Attribute**.

Map name is the name of the resulting map. It follows the selected input object until you type a name of your own, and it must not be empty when the map is built.

© 2026 Geomage 2003 LTD

Build parameters

- **Algorithm** — **ABOS**, **Kriging** or **Adaptive Snap**. The **Advanced** tab always shows the settings of the algorithm selected here.
- **Step X** and **Step Y** — the cell size of the resulting map, 25 m by default. These boxes, and every other distance in the dialog, are labelled and entered in the distance unit of the project, so an imperial project shows the same default in feet.
- **Map box** — the area the map covers: **Input points** (the extent of the input data), **All bins** (the whole survey, which requires a defined survey bounding box) or **Polygon**, which adds a **Polygon** list for choosing the outline.

Fault handling

Select in **Fault handling** how faults constrain the map:

- **Ignore faults** — faults are not used and the map is continuous across them. The remaining fault controls are hidden.
- **Use selected faults** — click **Select faults...** and tick the faults to be used as barriers. The label beside the button shows how many faults are ticked, or **All** when nothing is ticked and every fault of the project is used.
- **Use fault model** — use the surfaces of the model selected in **Fault model** as the barrier framework.
- **Fault polygon** — use the outlines of the polygon selected in **Fault polygon** as the barriers.

With every option except **Ignore faults** the **Faults clear radius** box appears. Input data lying closer to a barrier than this distance is discarded and that strip is rebuilt from the reliable data further away, which removes noisy values along the fault without leaving the strip empty.

Smoothing and trend

Post-mapping smoothing smooths the finished map, and **Step** sets the width of the smoothing window. The option is cleared by default.

Use trend builds the map on top of a regional surface: the map selected in **Trend map** is subtracted from the input data, the difference is interpolated and the trend is added back. The trend map must hold the same kind of values as the input data, so a depth input needs a depth trend map. Input points that the trend map does not cover are not used, and their number is reported when the map is built. The option is cleared by default.

© 2026 Geomage 2003 LTD

Advanced

The **Advanced** tab shows the settings of the algorithm selected on the **General** tab:

- **ABOS** — **Use nearest**, **Max error**, **Number of iterations**, **Factor**, **Taper size**, **Smooth aperture**, **Smoothing iterations** and **Smoothing Q**.
- **Kriging** — **Covariance** (**Spherical**, **Gaussian** or **Exponential**), **Range** and **Max points**.
- **Adaptive Snap** — **Fault mode** (**Ignore**, **Barrier** or **Final refinement only**) and **Density** (**Auto**, **Sparse (wells)** or **Dense (grid)**).

Building the map

- **Run** — builds the map and leaves the dialog open.
- **OK** — builds the map and closes the dialog. With no input object selected it only stores the parameters and closes.
- **Close** — closes the dialog without building a map.

The result is saved in the **Calculated maps** folder of the Data Manager 124. If a map of the same name is already there, you are asked whether to overwrite it.

See Also

Mapping 724

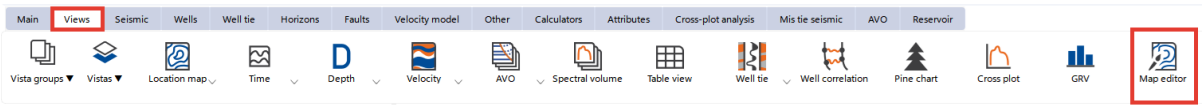
Map creating 727

Modelling 100

9.11.3 Map editing

Once the maps are generated, they can be edited using g-Space's Map Editor 341. The editor provides tools for refining, annotating, and enhancing your maps for presentations or further analysis.

In the Views bar 38, select the **Map Editor View** to open the map editing interface.

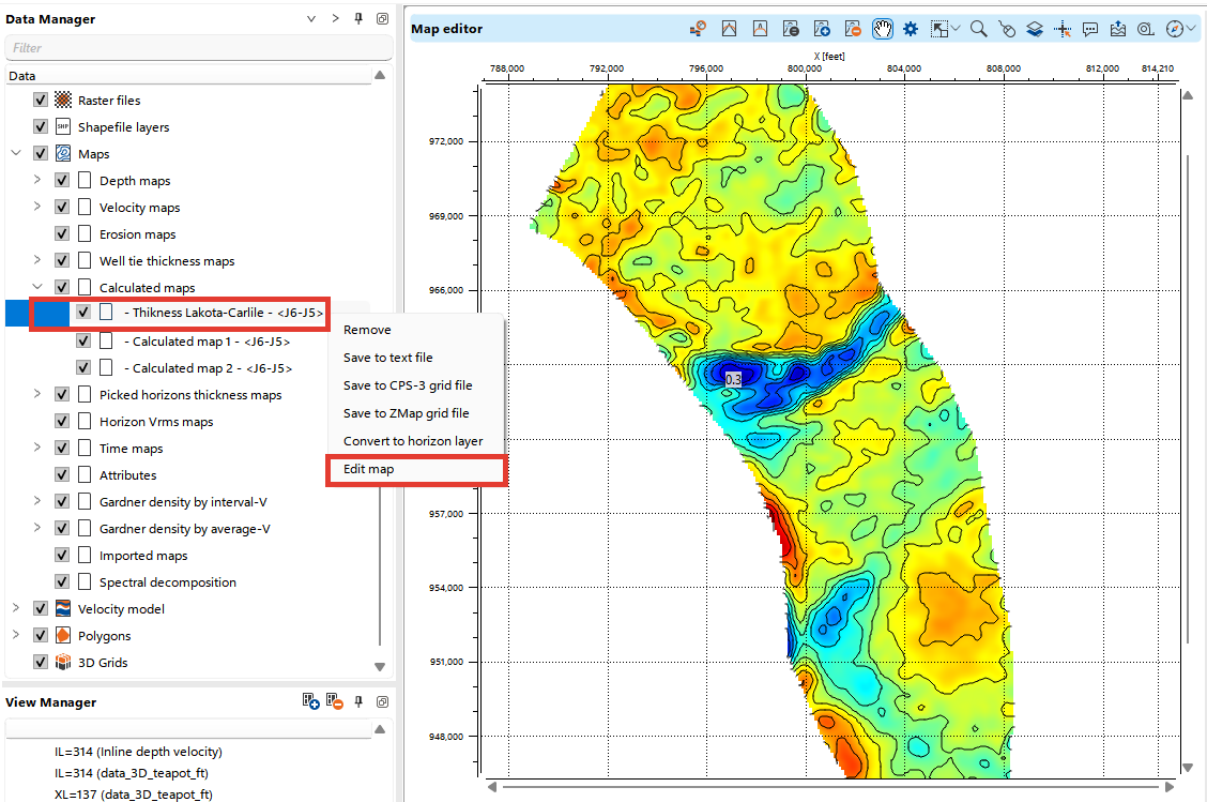


In the **Data Manager**, locate the map you want to edit.

© 2026 Geomage 2003 LTD

Right-click on the map and select **Edit map**. The selected map will then be displayed in the **Map Editor View**.

In the **Visual Settings** panel, adjust the visualization settings to customize the appearance of the map. This includes color scales, isolines, and other display options to better represent the data.



The full range of settings is available in the description of the Map editor view 341

Isoline labels

The **Visual Settings** panel provides dedicated settings for controlling how contour (isoline) labels are displayed on the map:

- **Show isolines labels** — selects which contours are labelled: **No labels**, **Major only** or **All**.
- **Isoline labels distance** — the spacing, in project distance units, at which labels are repeated along each contour.
- **Isolines labels size** — the text size of the labels.
- **Isolines labels color** — the colour of the label text.
- **Rotate isolines labels** — aligns each label with the direction of the contour it annotates. This option is selected by default, so isoline labels follow the contour direction on newly displayed maps; clear it to keep the labels horizontal.

© 2026 Geomage 2003 LTD

- **Isoline labels top faces** — sets which side of the contour the top of the label faces: **Smaller values (depth, time)** or **Larger values (thickness, isochore)**.

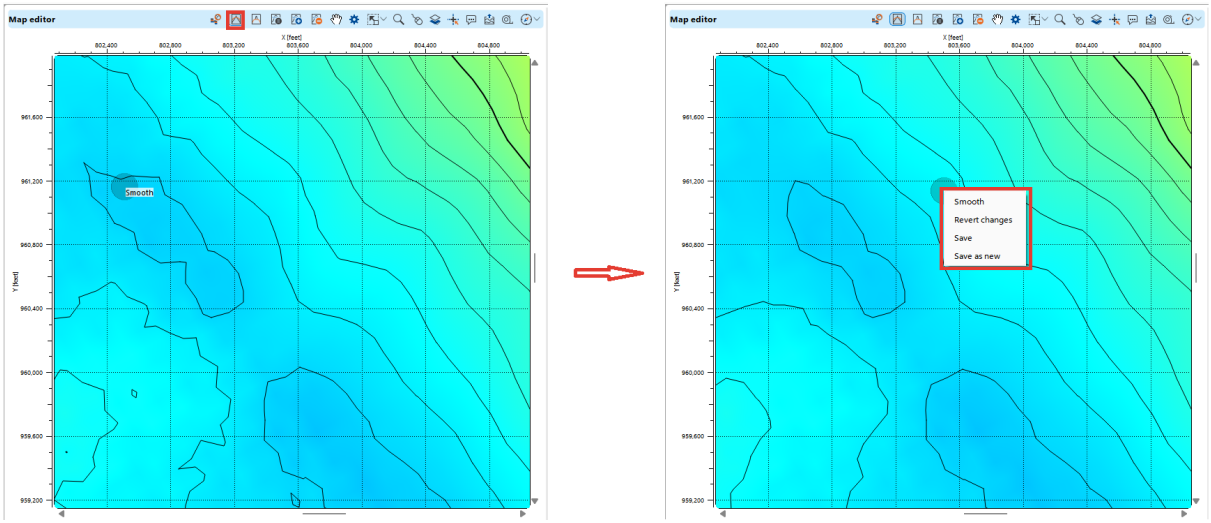
Labels are positioned automatically. The contour is interrupted where its value is printed, so the line does not run through the digits, and each label keeps a small clear margin around it. Labels are also skipped where a contour comes close to the edge of the map or to the edge of the data, so contours that only run along the boundary can stay unlabelled.

Set **Isoline labels top faces** to match the map type. **Smaller values (depth, time)**, the default, turns the top of each label toward the smaller contour values, which is the structural high on depth and time maps; **Larger values (thickness, isochore)** turns it toward the larger values, which is the thickest area on isochore, isopach and isochron maps. With **Isoline labels distance** set to 0 each contour is labelled once near its middle; enter a positive distance to repeat the label along the contour at that spacing.

Below are some practical examples of using map editing tools.

The **Smooth** tool is designed to reduce sharp gradients, noise, or irregularities in map data, making it more continuous and geologically realistic.

To undo changes, press **Alt** and right-click the mouse. A menu will appear, providing options to revert the changes or proceed with other actions.



Notes

Depth maps generated from seismic horizons can sometimes exhibit small, unnatural variations caused by interpolation or gridding.

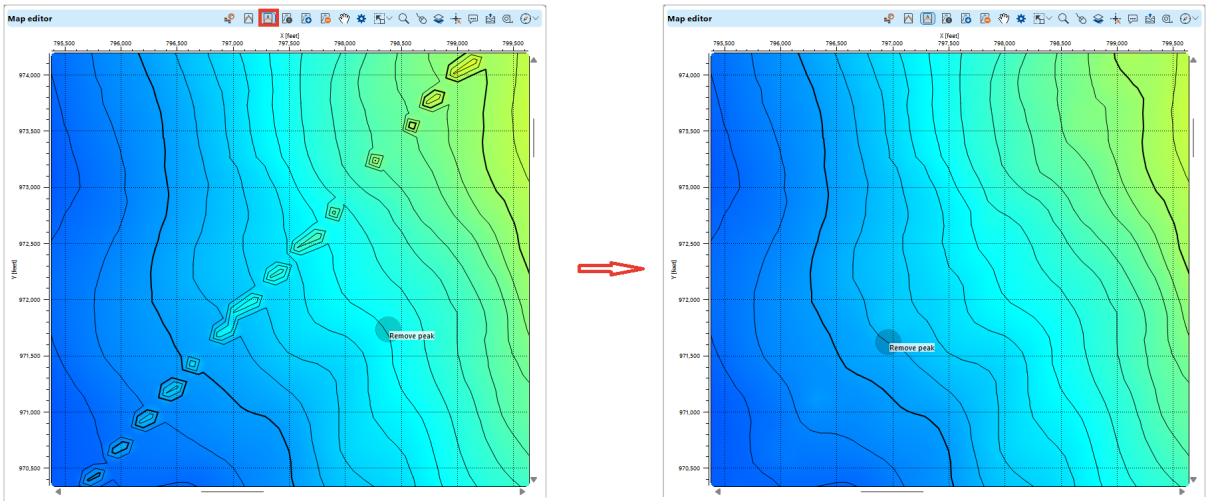
Velocity models often have abrupt changes between zones that may not accurately reflect the gradual transition of rock properties.

© 2026 Geomage 2003 LTD

Faults may introduce sharp breaks or offsets in horizon maps.

After gridding well, seismic, or attribute data, sharp transitions may occur due to data gaps or uneven distribution.

The **Remove Peak** tool is particularly useful in geophysical and geological workflows where small, isolated, or anomalous high points (peaks) in the data may distort interpretation or modeling.



Notes

Seismic attribute maps, such as amplitude or frequency maps, may occasionally contain small, spurious peaks due to noise or localized data inaccuracies.

In structural geology, isolated peaks in depth or time maps may not represent true geological structures but rather noise or interpolation artifacts.

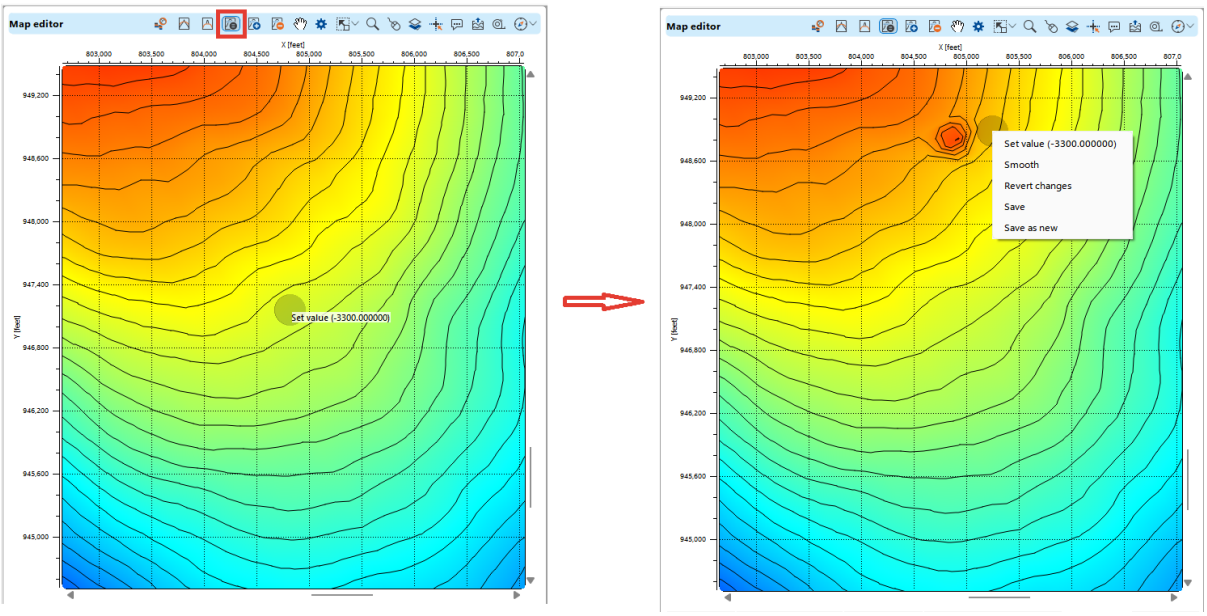
For large regional studies, maps may exhibit peaks that are not representative of the broader geological trend, often due to noise, small-scale structures, or surface irregularities.

In velocity modeling, spurious velocity spikes may cause errors in depth conversion and lead to misinterpretations of subsurface features.

The **Set value** tool allows you to assign a specific value to any part of the map. This is useful for correcting or standardizing certain areas of the map where the data may not accurately represent the real geological or geophysical structure.

To set a particular value or revert changes, press **Alt** and right-click the mouse. A menu will appear, providing options.

© 2026 Geomage 2003 LTD



The **Add value** and **Subtract value** tools allow you to increase or decrease the values at any part of the map by a specified amount, facilitating data adjustments; after selecting the tool, you can define the amount you want to add or subtract from the current map values by pressing **Alt** and **right-clicking** the mouse, at which point a menu will appear, providing options.

Creating, editing and saving faults in Map editor View is described in detail in the Faults 627 chapter.

9.11.4 Residual calculation

Residual calculation compares a map with marker values at well locations and reports the difference (the residual) for every well. Use it to check how closely a structural map honours the well data, to find the wells where the map is too high or too low, and to keep the result for further analysis.
For each well in the selected scope the wizard takes the well-path position of the marker, samples the map at that position, reads the selected marker attribute and calculates the residual with the chosen formula. The results are shown in the wizard as summary statistics and a table, and can be stored as a point set in the Points 185 branch of the Data manager 124.

Requirements

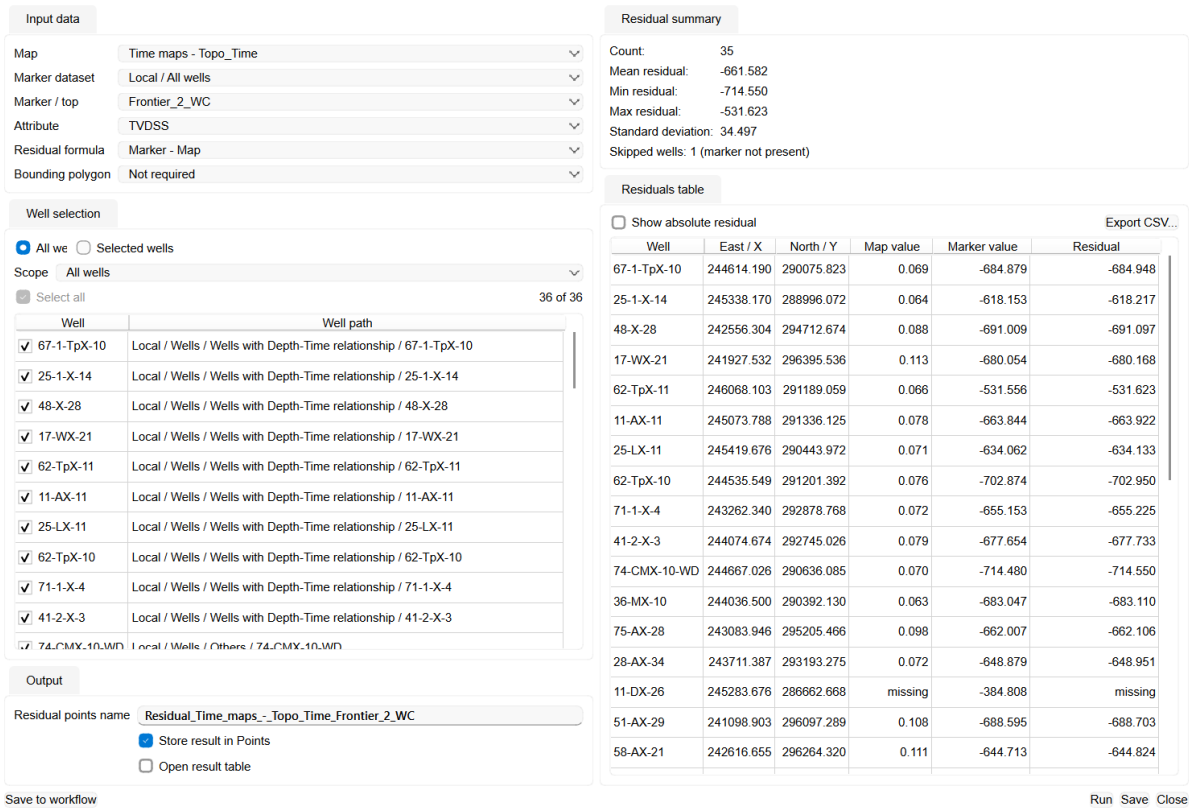
© 2026 Geomage 2003 LTD

- At least one map in the project. **Internal topography** and **External topography** can also be used as the map when they are loaded.
- At least one marker set or zone set that contains picked markers.
- Wells in which the selected marker is picked. Wells without that marker are reported and produce no result row.

Opening the wizard

Open the wizard from the Modelling 100 ribbon tab → **Map calculator** group → **Residual calculation**.
You can also start it from the map you want to check: in the Data manager 124, right-click a map under Maps 162 and choose **Calculate residuals with markers**. That map is then already selected in the wizard.

The wizard opens as one dialog. The **Input data**, **Well selection** and **Output** groups are on the left, the **Residual summary** and the **Residuals table** are on the right, and the **Save to workflow**, **Run**, **Save** and **Close** buttons are along the bottom.



© 2026 Geomage 2003 LTD

Input data

- **Map** — the map to check. Project maps are listed as map type followed by map name, for example `Time maps - Topo_Time`.
- **Marker dataset** — the marker or zone set to compare the map with. Entries are listed as user and set name, and zone sets are marked **(Zones)**.
- **Marker / top** — the marker (or zone) inside the selected dataset.
- **Attribute** — which marker value to compare with the map: **TVDSS**, **TVD**, **MD** or any custom attribute defined for that marker. For a zone set only **TVD** is offered.
- **Residual formula** — **Marker - Map** (default) subtracts the map value from the marker value; **Map - Marker** is the opposite sign.
- **Bounding polygon** — **Not required** (default), or one of the project polygons 167 to limit the calculation to an area. Wells outside the polygon are left out of the result.

Well selection

The **Well selection** group defines which wells take part in the calculation. The list shows a **Well** column with the well name and a **Well path** column with its position in the Data manager tree.

- **Scope** — the group of wells to work with: **All wells**, a well filter (**Filter:** and the filter name), a Data manager well folder (**Folder:** and the folder name) or a well section (**Section:** and the section name). Choosing a scope marks the wells that belong to it in the list.
- **All wells** — the default. Every well of the selected scope is used and the list is read-only.
- **Selected wells** — makes the list editable so that you can tick individual wells. **Select all** ticks or clears all of them, and the counter to the right of it shows how many of the listed wells are ticked.

Output

- **Residual points name** — the name of the resulting point set. g-Space suggests `Residual_<map>_<marker>`, with spaces and slashes replaced by underscores, and keeps the suggestion in step with your selections until you type a name of your own.
- **Store result in Points** — on by default. The residuals are written as a point set in the **Points** branch when you click **Save**. Turn it off to review and export the residuals without creating a point set.
- **Open result table** — off by default. When it is on, the new point set becomes the active table 285 after saving. It applies only together with **Store result in Points**.

© 2026 Geomage 2003 LTD

Running the calculation

- Click **Run** to calculate without saving anything. The **Residual summary** and the **Residuals table** are filled with the computed values, so you can change the selections and run again.
- Click **Save** to calculate and keep the result. If a point set with the same name already exists, g-Space asks whether to replace it. If **Store result in Points** is off, a message reports how many wells were calculated and no point set is created.
- Click **Close** to leave the wizard without saving.

The **Run**, **Save** and **Save to workflow** buttons stay disabled until every selector has a value, at least one well is selected, and a residual points name is entered when the result is to be stored.

Residual summary

- **Count** — the number of rows in the residuals table.
- **Mean residual**, **Min residual**, **Max residual** and **Standard deviation** — statistics of the calculated residuals. Rows without a residual are ignored, and all four show – when nothing could be calculated.
- **Skipped wells** — appears when the selected marker is not picked in some of the wells; those wells produce no row. Point at the line to see their names. They are also listed in a message after **Save**.

Residuals table

The table has the columns **Well**, **East / X**, **North / Y**, **Map value**, **Marker value** and **Residual**. Enable **Show absolute residual** to add an **Absolute residual** column.

- Where the map has no value at the well position, **Map value** and **Residual** show `missing`. The row is kept in the table and in the saved point set, but it does not contribute to the summary statistics.
- Click a column header to sort the table. Rows without a value always stay at the end, whichever direction you sort in.
- Select cells and press **Ctrl+C** to copy them; the column headers are included when the whole table is selected.
- Click **Export CSV...** to write the table to a comma-separated (`.csv`) or tab-separated (`.tab`) file. Run the calculation first, otherwise there is nothing to export.

© 2026 Geomage 2003 LTD

The residual point set

With **Store result in Points** on, **Save** creates a point set under **Points** with one point per calculated well, placed at the well-path position of the marker. Each point carries the following attributes:

- **Well name** and **Marker name** — the well and the marker the residual belongs to.
- **Marker value**, **Map value** and **Residual value** — the compared values and their difference.
- **Residual formula used** — the formula applied to the point, or `missing` where the map had no value at that position.

Residual value is already assigned as the value attribute of the set, so the points are colour-coded by residual wherever point colouring by value is used. Otherwise the set behaves like any other Points 185 object: display it on the Location map 235 over the map you checked, open it in the Table view 285, or export it.

Save to workflow

Click **Save to workflow** to store the current settings as a **Residual calculation** job in the current workflow 535 instead of calculating now. The job keeps the map, marker dataset, marker, attribute, residual formula, bounding polygon, well selection, output name and output options, and it calculates the residuals and writes the point set when the workflow is executed.
Reopening a saved job restores those settings in the wizard, and the buttons change to **Save** and **Cancel**. If a saved selection no longer exists in the project, that selector is left empty and a warning under **Input data** names what is missing; choose replacements before saving, because the job cannot be saved or run while a reference is unresolved.

See Also

- Map creating 727
- Map editing 733
- Modelling 100
- Points 185
- Working with Markers 551
- Table view 285
- Workflows 535

© 2026 Geomage 2003 LTD

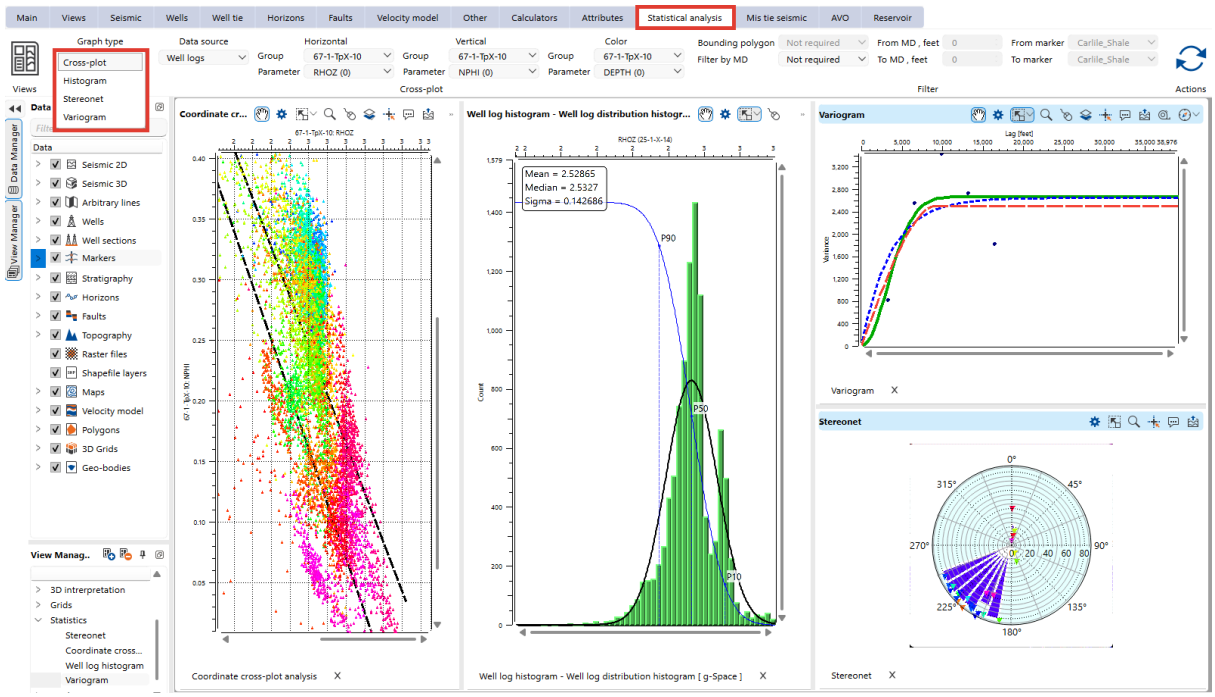
9.12 Statistical analysis

The **Statistical Analysis** view provides a robust toolkit for analyzing and visualizing relationships between well log attributes, data distributions, and structural orientations.

By selecting the appropriate **Graph Type** — Cross-Plot 745, Histogram 748, Variogram, or Stereonet 756 — users can gain valuable insights into their data through targeted visualization techniques.

For effective analysis, users can fine-tune graph settings such as **data source selection**, **axis parameters**, and **filters** to focus on specific intervals or attributes of interest. The use of **Bounding Polygons**, **MD ranges**, and **markers** allows for precise control over the data included in the analysis, enabling more accurate and meaningful results.

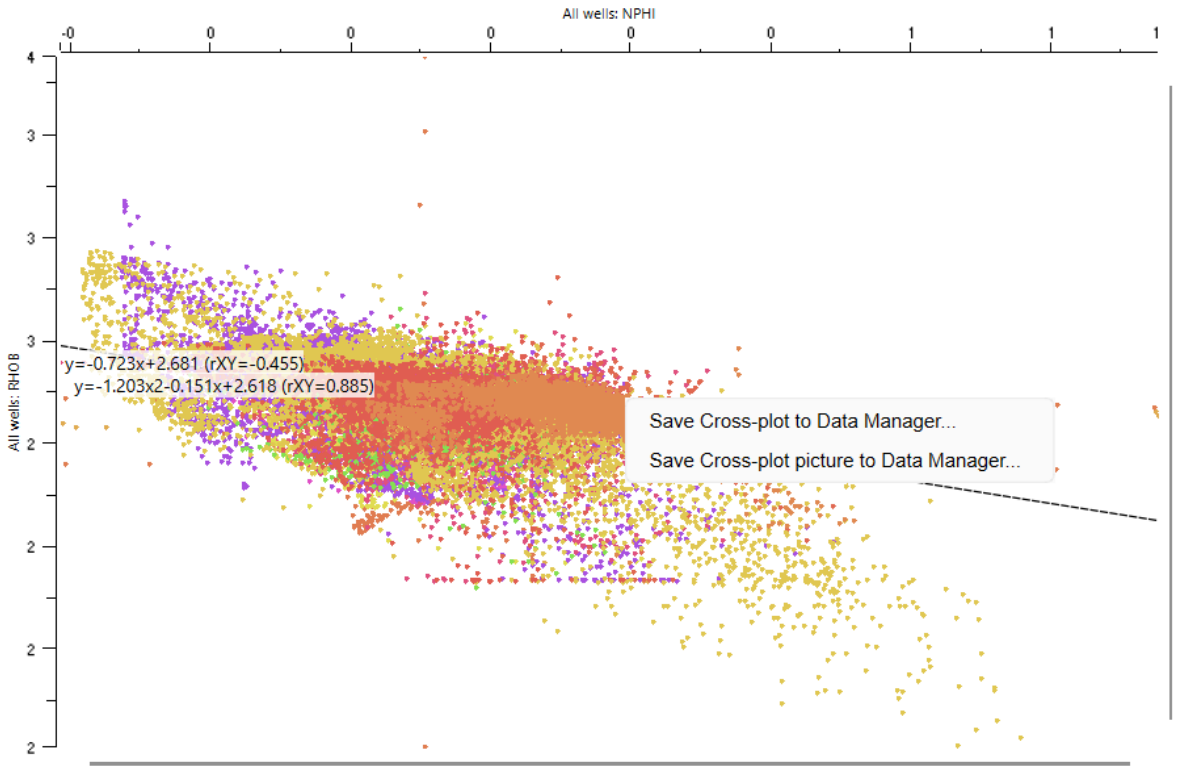
Visual customization is an essential part of the workflow. Adjusting **color schemes**, **point size**, and **bin settings** helps improve clarity and enhances the interpretability of graphs.



Saving graphs to the Data Manager

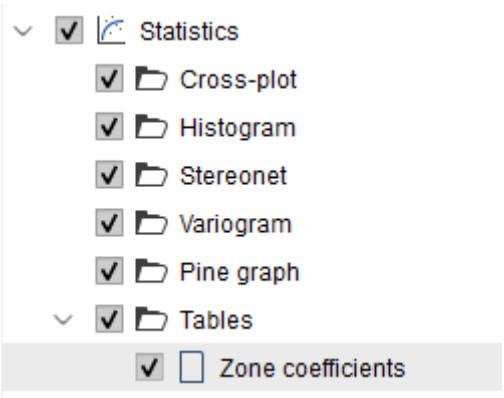
A graph can be stored in the project, so that it can be reopened later without setting up its parameters again. Right-click the plot in the **Cross plot**, **Histogram**, **Stereonet** or **Variogram** view and choose the save command of the current graph type, for example **Save Cross-plot to Data Manager...**. Type a **Name** for the new object and confirm. The name must not be already used in the same folder.

© 2026 Geomage 2003 LTD



In the **Cross plot** view the context menu also offers **Save Cross-plot picture to Data Manager...**, which stores a static picture of the plot as it currently looks instead of its settings.

Saved graphs are collected in the **Statistics** category of the Data manager 124, which contains a folder for each graph type: **Cross-plot**, **Histogram**, **Stereonet**, **Variogram** and **Pine graph**, together with a permanent **Tables** folder for project tables. The **Pine graph** folder is filled from the Pine graph view 332. A saved cross-plot additionally lists the equations of its linear and quadratic regression lines as separate records inside the object.



Right-click a saved item in the Data manager to:

© 2026 Geomage 2003 LTD

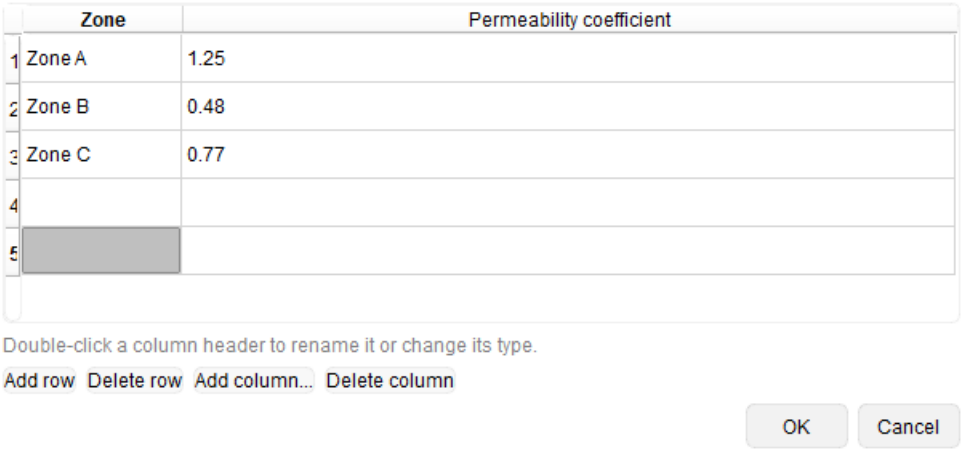
- **Open** – reopens the saved item. A graph saved from the Statistical analysis view is restored with its graph type, data source, groups, parameters and filters, and that view is brought to the front; a saved picture is shown in a separate window.
- **Rename** – gives the item another name.
- **Delete** – removes the item from the project.

A regression record only offers **Delete**: it keeps the equation of the line as it was when the cross-plot was saved.

Project tables

The **Tables** folder of the **Statistics** category keeps tabular data as project objects, for example permeability coefficients per zone, petrophysical cutoffs or core plug measurements. A table is a generic object: it can combine text and numeric columns in any order, it may describe one well, several wells or the whole project, and columns such as a well name, a depth or a zone are used only when the data needs them.

To create a table, right-click the **Tables** folder and choose **New table**. Type a **Name** that is not used by another table in the folder and confirm. The table editor opens on a starter grid of two columns and five empty rows and carries the table name as its title.



Type the values directly into the cells, and shape the table with the buttons under the grid:

- **Add row** – adds an empty row at the end of the table.
- **Delete row** – removes the row of the selected cell.
- **Add column...** – asks for the **Name** of a new column and for its **Type**, **Text** or **Number**.
- **Delete column** – removes the column of the selected cell.

Double-click a column header to rename that column or to change its type. **OK** stores the table in the project; when a cell of a **Number** column holds a value that is not a number, the editor names that column and row so that the value can be corrected first.

© 2026 Geomage 2003 LTD

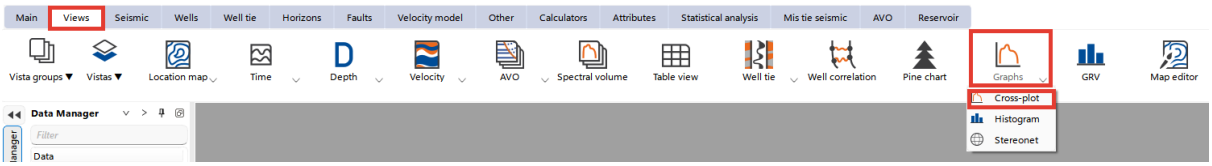
Data prepared outside the project can be brought in instead: right-click the **Tables** folder, choose **Import table from file** and select a **Delimited text (*.csv *.txt *.tsv *.dat)** file. Answer whether the first row of the file holds the column names, then type a name for the new table; the file name is offered as the default.

Right-click a saved table to **Open** it in the editor, to **Rename** it or to **Delete** it from the project. An open table is written back to the project when the editor is closed with **OK**.

9.12.1 Cross plot

Cross-plot analysis is allowing users to compare two variables on a scatter plot to identify relationships, trends, or anomalies.

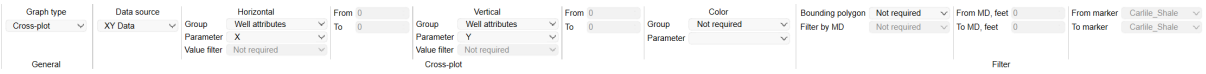
To create cross plot view go to the Views bar 38, navigate to **Graphs** and click on **Cross plot** button:



To set what data to be analyzed set parameters in the Statistical analysis 111 bar:

Cross-plot analysis is helpful to the user to understand the relation between two or more seismic attributes. In this cross-plot analysis bar the user can able to select the seismic attributes and generate the cross plot in Views

In the **Data Source** dropdown menu, you can choose the type of data to analyze. Available options include **XY Data**, **Well Logs**, and **Markers**.



When **XY Data** is selected as the data source in the **Cross-Plot Analysis** tool, the interface allows users to analyze relationships between two variables (X and Y coordinates). This mode is particularly useful for investigating well attributes, seismic attributes, or any spatially correlated data that can be expressed in an XY format.

For the **Horizontal Group (X-axis)** and **Vertical Group (Y-axis)**, you can select a group of well attributes (such as X, Y, KB), markers, topography, grid maps, or horizons as the group for analysis. You can then specify the desired parameter from each group.

© 2026 Geomage 2003 LTD

The **Color** option allows you to define a third parameter for analysis, which will be used to color the data points on the cross-plot. This additional dimension provides deeper insight into data distribution or behavior. For example, you could color the data points based on a seismic attribute like amplitude or porosity, adding another layer of interpretation.

The **Bounding Polygon** option allows you to limit the analysis to specific areas of interest. By defining a polygon, you can filter out data points outside the selected region. If no polygon is required, leave this option set to **Not required**.

When **Well Logs** is selected as the data source in the **Cross-Plot Analysis** tool, the interface enables the analysis of relationships between well log measurements for individual wells. This is especially useful for examining the correlation between different well log parameters (e.g., porosity, resistivity, sonic log) within a specific well.

For the **Horizontal Group (X-axis)** and **Vertical Group (Y-axis)**, you can select a specific well from the **Group** dropdown menu and then specify the parameter from the selected well (such as **RHOZ**, **DEPTH**, **DT**, etc.).

The **Group** dropdown is not limited to a single well. In addition to any individual well, you can choose **All wells** to analyze the log data of every well at once, or a named **well section** to analyze a saved group of wells (well sections act as folders/filters of wells). The parameter list then contains the log curves shared by the wells in scope. These options are available for the Horizontal and Vertical groups.

The **Color** option enables you to add a third parameter to your analysis. This third parameter is used to color the data points in the cross-plot, allowing you to visualize how a specific log (such as sonic log **DT**) varies across the plotted data.

In **Well logs** mode the **Color** column contains the **Parameter** dropdown only: there is no Color **Group** dropdown, and the coloring always covers every well that has log data. The list starts with **Wells** and **No color**, followed by the log curves available for the wells in scope. Choose **No color** to leave the points in a single default color.

Besides a log curve, the **Color** parameter can be set to **Wells**. In this mode the points of each well are drawn in their own distinct color and each well is added to the legend as a separate entry labeled with the well name, so you can tell the wells apart on the chart. **Wells** is listed as soon as at least one well with log data is loaded, and it is the value already selected for the **Color** parameter on a new cross-plot.

© 2026 Geomage 2003 LTD

Filter by MD

- **Manual**: Allows you to manually input the MD values.
- **By markers**: Uses predefined geological markers to select the interval of the analysis.

Filter by value

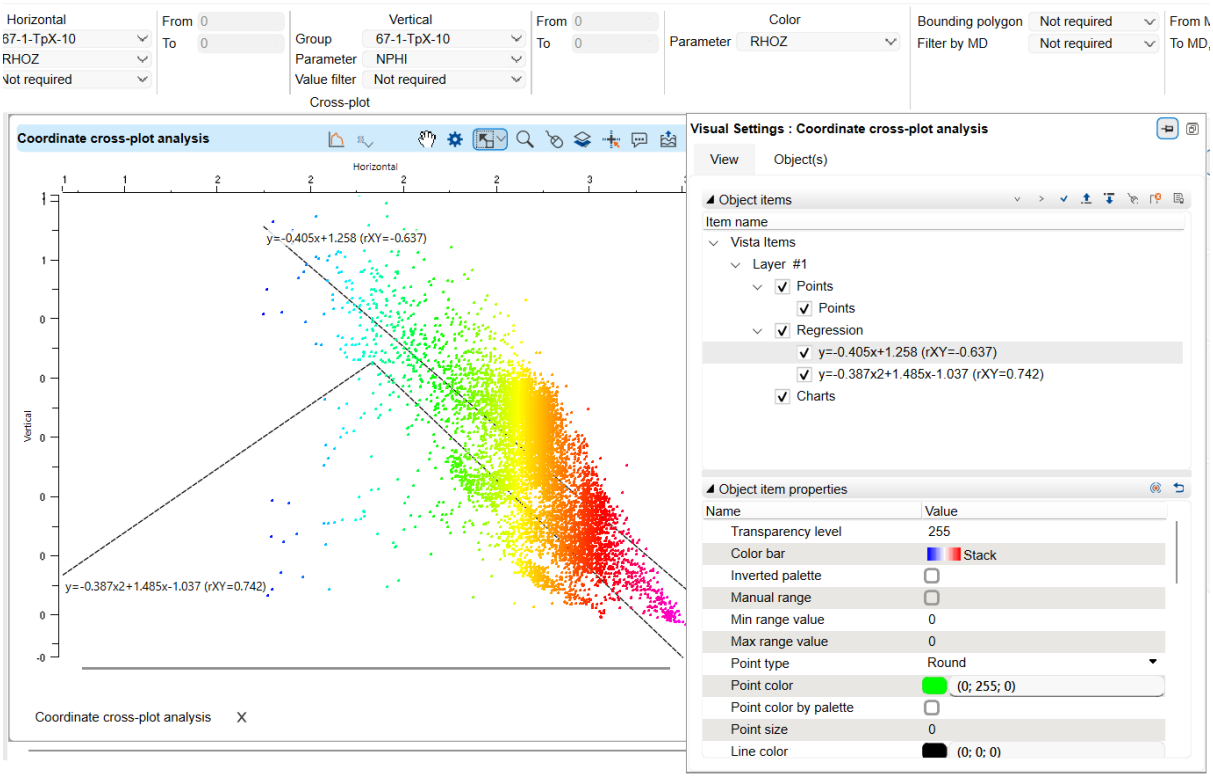
For the **Well Logs** data source you can also restrict the analysis to a value range on each selected parameter. A **Value filter** dropdown is provided for the Horizontal and Vertical parameters, below the corresponding **Parameter** dropdown, with its **From** and **To** fields in the next ribbon column:

- **Horizontal value filter** — From / To
- **Vertical value filter** — From / To

Set a filter to **Manual** and enter the **From** and **To** bounds of the range to keep; samples whose value falls outside the range are dropped. Leave a filter set to **Not required**, the default, when no value range is needed. The value filters can be edited only when **Data source** is set to **Well logs**; in **XY Data** and **Markers** mode the rows stay visible but are greyed out. Value filters are applied together with the MD, marker and bounding polygon filters, and the cross-plot is refreshed using the filtered log samples only.

In the **Cross-plot** view, you will find two regression functions: **linear** and **quadratic**. These functions help in identifying trends and correlations between the selected parameters. Both regression lines are also accessible through the **Object** tab within the **Visual Settings** panel, where you can further customize the visual settings, such as line color, thickness, and other aesthetic preferences, to suit your needs.

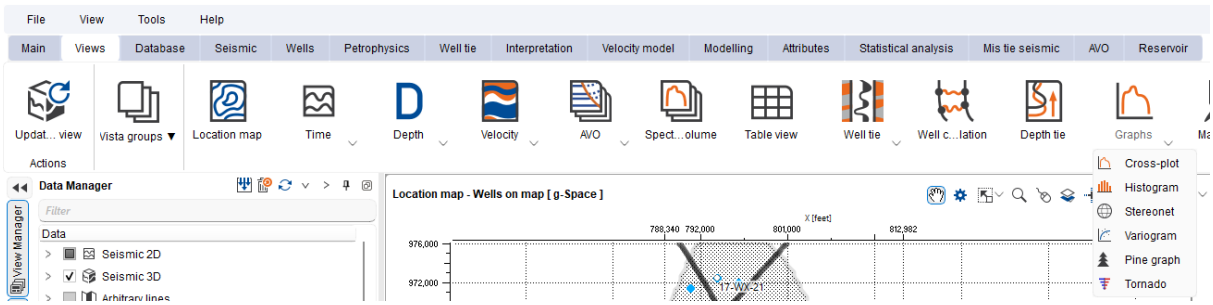
© 2026 Geomage 2003 LTD



9.12.2 Histogram

Histogram analysis allows users to visualize the distribution of a single variable, helping to identify frequency patterns, ranges, and data trends.

To create Histogram view go to the Views bar 38, navigate to **Graphs** and click on **Histogram** button:



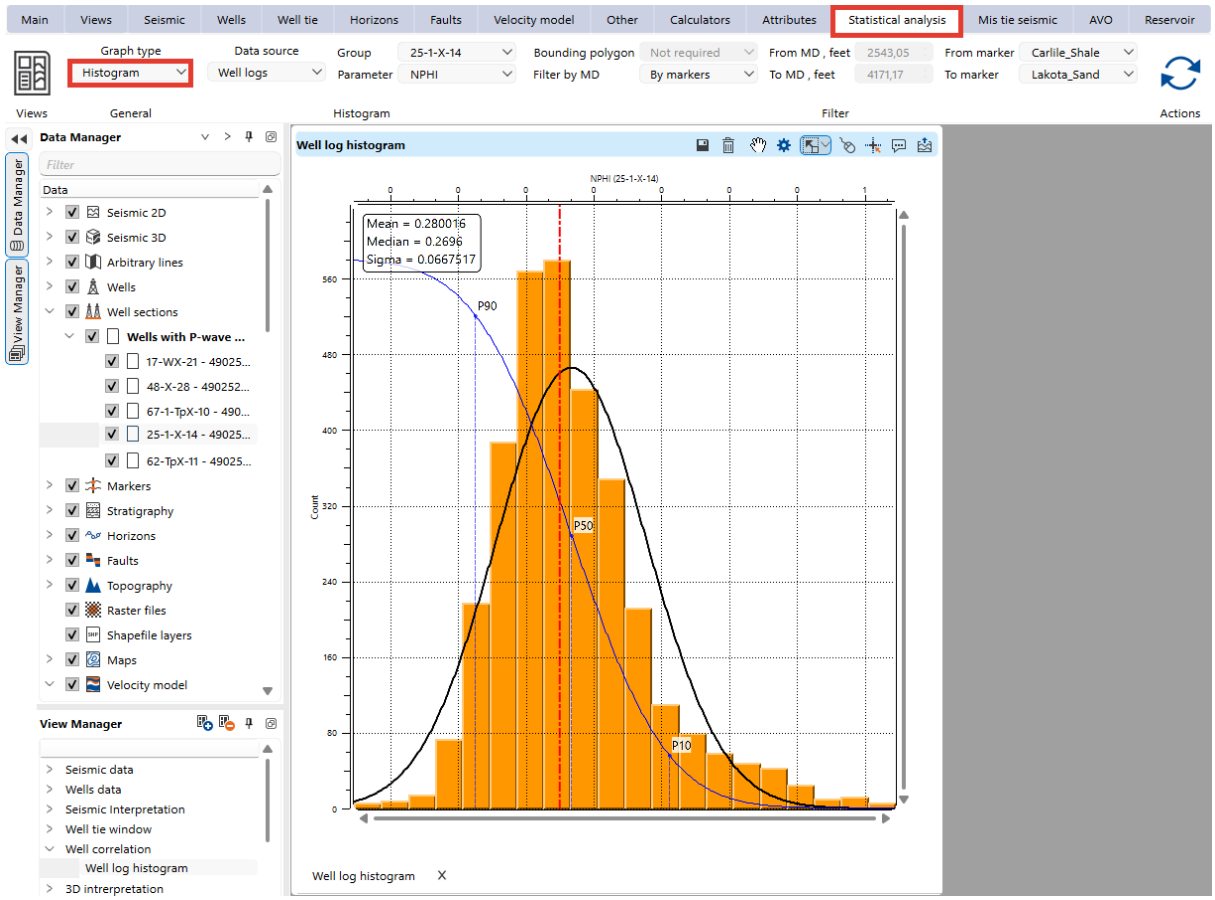
© 2026 Geomage 2003 LTD

Each click on **Histogram** opens a new **Well log histogram** view, so several histograms can be kept open at the same time. A new view starts with the parameters of the histogram view that is active at that moment.

To configure the analysis, set the parameters in the Statistical Analysis 742 bar:

- **Data Source**: Available options include **Well Logs** and **XY Data**.
  - **Well Logs**: Select a specific well and choose a parameter such as **RHOZ**, **DT**, or any other well log measurement.
  - **XY Data**: This mode allows users to analyze numerical values, such as seismic attributes or spatial data, represented in XY format.
- **Group and Parameter**: When working with **Well Logs**, users can specify the desired well log group and the parameter to be analyzed. For **XY Data**, users can select a group containing numerical datasets (e.g., amplitude, porosity).
- **Filter Options**: Use **Bounding Polygon**, **MD Range**, or **By Markers** to refine the data included in the analysis:
  - **Filter by MD** allows you to define the depth interval manually or through predefined markers.
  - **Bounding Polygon** restricts the analysis to specific areas of interest.

© 2026 Geomage 2003 LTD

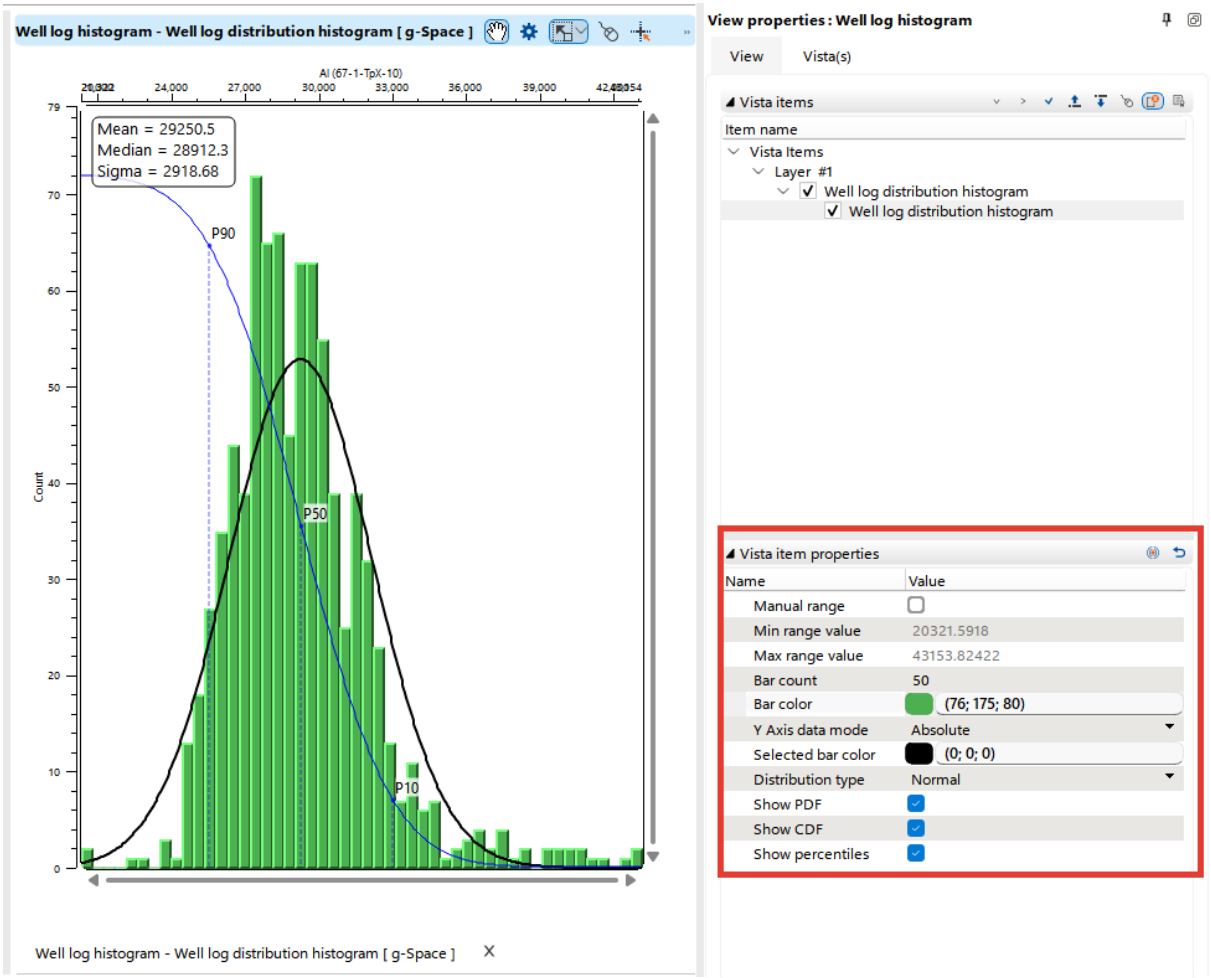


These parameters belong to the active **Well log histogram** view only. Changing a parameter updates that view alone, while the other opened histogram views keep their own parameters and the distribution they already display. When another histogram view is activated, the **Statistical analysis** bar shows the parameters saved for that view.

The **Histogram** view automatically displays the following statistical parameters for the analyzed data:

- **Mean**
- **Median**
- **Standard Deviation**

© 2026 Geomage 2003 LTD



Visual Settings

Users can customize the **Histogram** display for better interpretation:

- **Bar Count**: Adjust the number of bars to control the resolution of the histogram.
- **Bar Color**: Set the color of the bars for improved visual clarity.

Optional overlays can be added to the histogram:

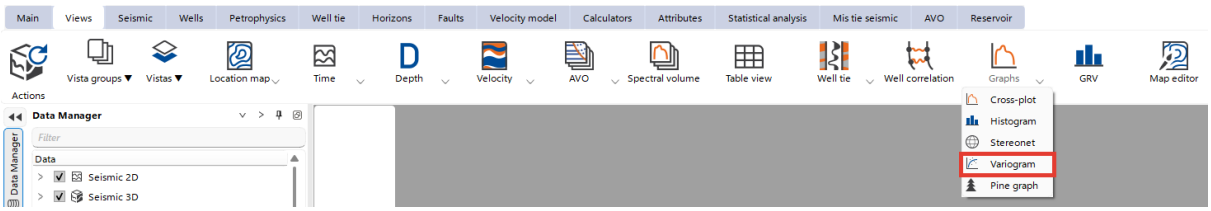
- **Probability Density Function (PDF)**: Visualizes the likelihood of data values within the distribution.
- **Cumulative Distribution Function (CDF) Curve**: Represents the cumulative frequency of the dataset.
- **Percentile Markers**: Visualize key percentiles, such as **P90**, **P50**, and **P10**, for additional statistical insights.

© 2026 Geomage 2003 LTD

9.12.3 Variogram

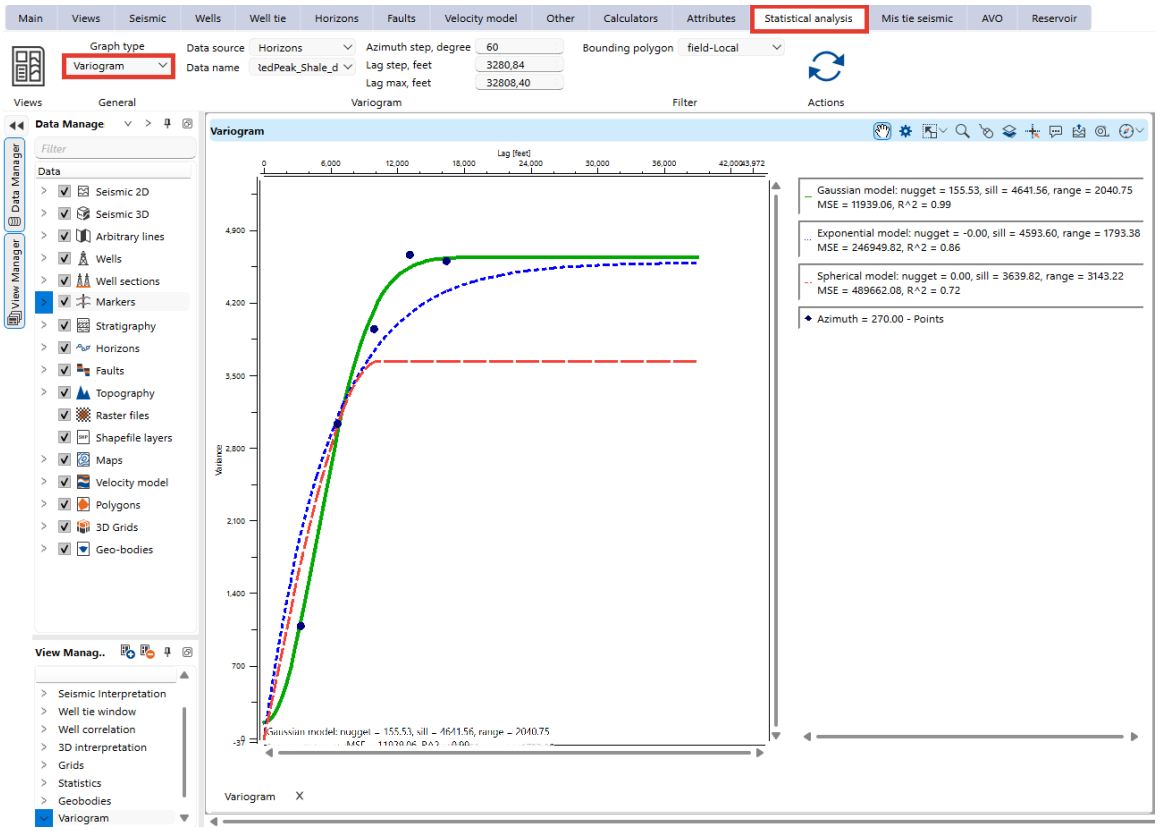
Variogram analysis is a crucial statistical tool used in geoscience to measure spatial variability and correlations between data points. This chapter explains how to perform variogram analysis using the provided tools and interface, illustrated with images for clarity.

To create Variogram view go to the Views bar 38, navigate to **Graphs** and click on **Variogram** button:



To start variogram analysis, navigate to to the Statistical Analysis 742 bar and select the **Variogram** option under the Graph type dropdown menu.

© 2026 Geomage 2003 LTD



Data Sources for Variogram Analysis

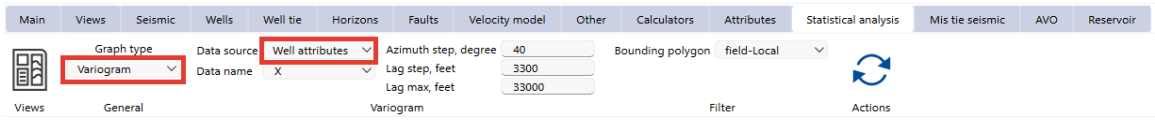
1. Well Attributes

Select "Well Attributes" as the data source from the dropdown menu.

Choose the attribute (X, Y, Total depth, KB or Ground level) from the Data Name field.

Specify parameters:

- Azimuth step
- Lag step
- Maximum lag distance



2. Well Logs

Choose "Well Logs" as the data source.

Select a specific well log Data Name field.

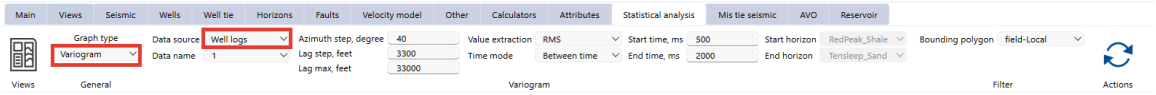
© 2026 Geomage 2003 LTD

Specify parameters:

- Azimuth step
- Lag step
- Maximum lag distance

Additional settings include:

- Value extraction method (Average, RMS, Minimum, Maximum, Most common, Median)
- Range for analysis - between depths or horizons
- Bounding polygon for spatial filtering.



3. Markers

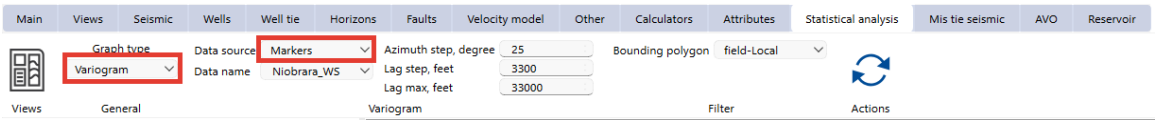
Select "Markers" as the data source.

Choose a geological marker from the Data Name dropdown menu.

Define azimuth step, lag step, and maximum lag distance for spatial analysis.

Additional settings include:

- Bounding polygon for spatial filtering.



4. Grid maps

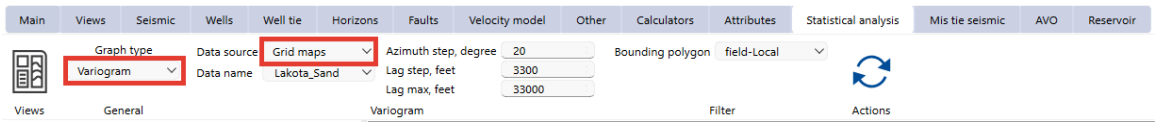
Select "Grid maps" as the data source.

Choose a map from the Data Name dropdown menu.

Define azimuth step, lag step, and maximum lag distance for spatial analysis.

Additional settings include:

- Bounding polygon for spatial filtering.



© 2026 Geomage 2003 LTD

5. Horizons

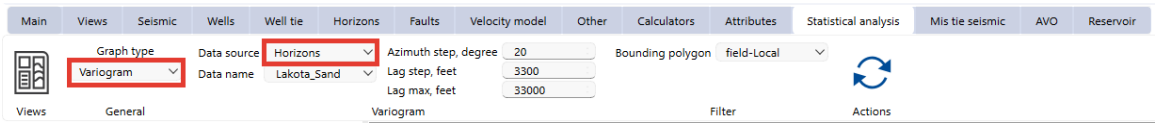
Select " Horizons" as the data source.

Choose a horizon from the Data Name dropdown menu.

Define azimuth step, lag step, and maximum lag distance for spatial analysis.

Additional settings include:

- Bounding polygon for spatial filtering.



Once you generate a variogram, it appears in the **Variogram view**. The plot displays experimental variograms as points and fitted models as curves.

Fitted Models:
The tool provides three common variogram models:

Spherical

Exponential

Gaussian

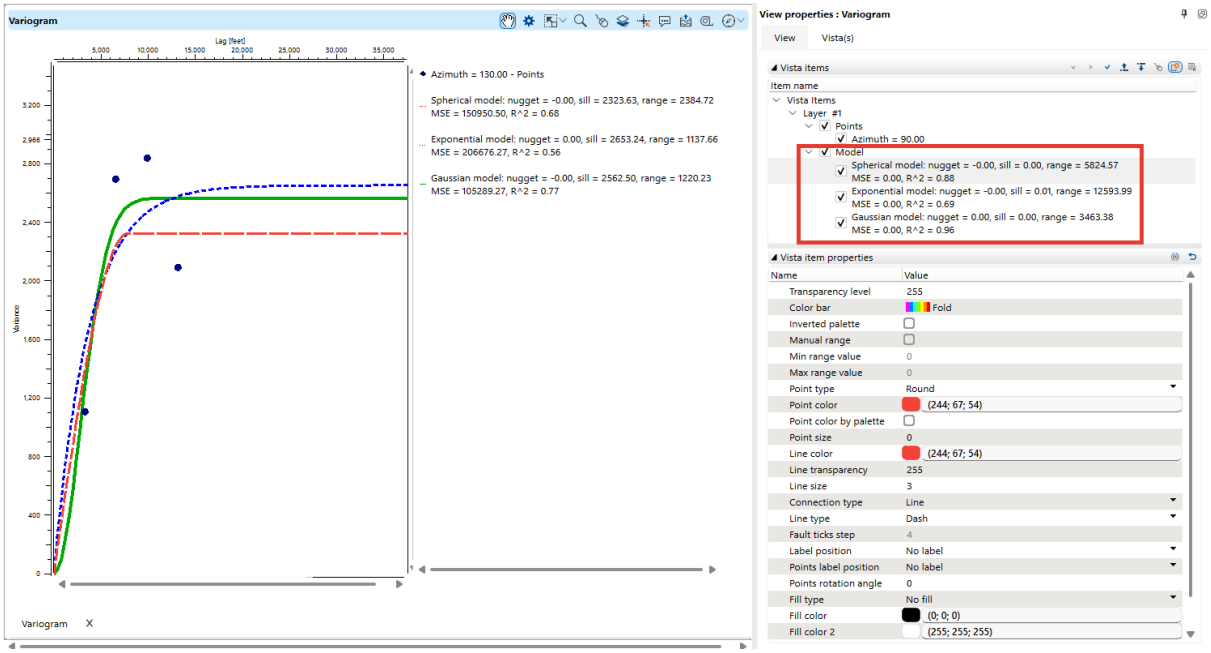
Each model is characterized by:

Nugget: Represents measurement error or micro-scale variation.

Sill: The total variance where the model flattens out.

Range: The distance at which spatial correlation diminishes.

© 2026 Geomage 2003 LTD



In the Visual Settings panel, users can adjust visual settings for better interpretation:

- Transparency levels
- Color schemes
- Line types and thickness
- Point size and shape

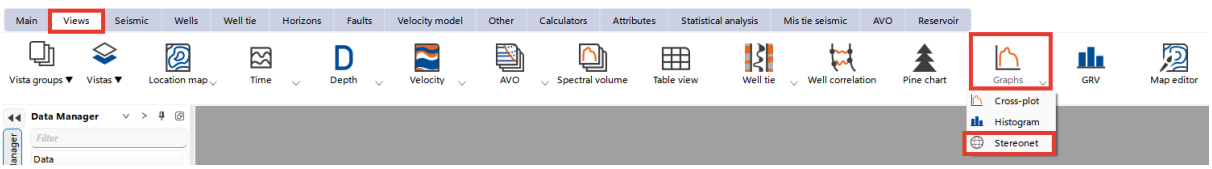
Users can also enable or disable specific models to focus on relevant data trends.

9.12.4 Stereonet

Stereonet analysis is a powerful tool for visualizing structural data, such as dip and azimuth measurements, to analyze geological orientations and patterns.

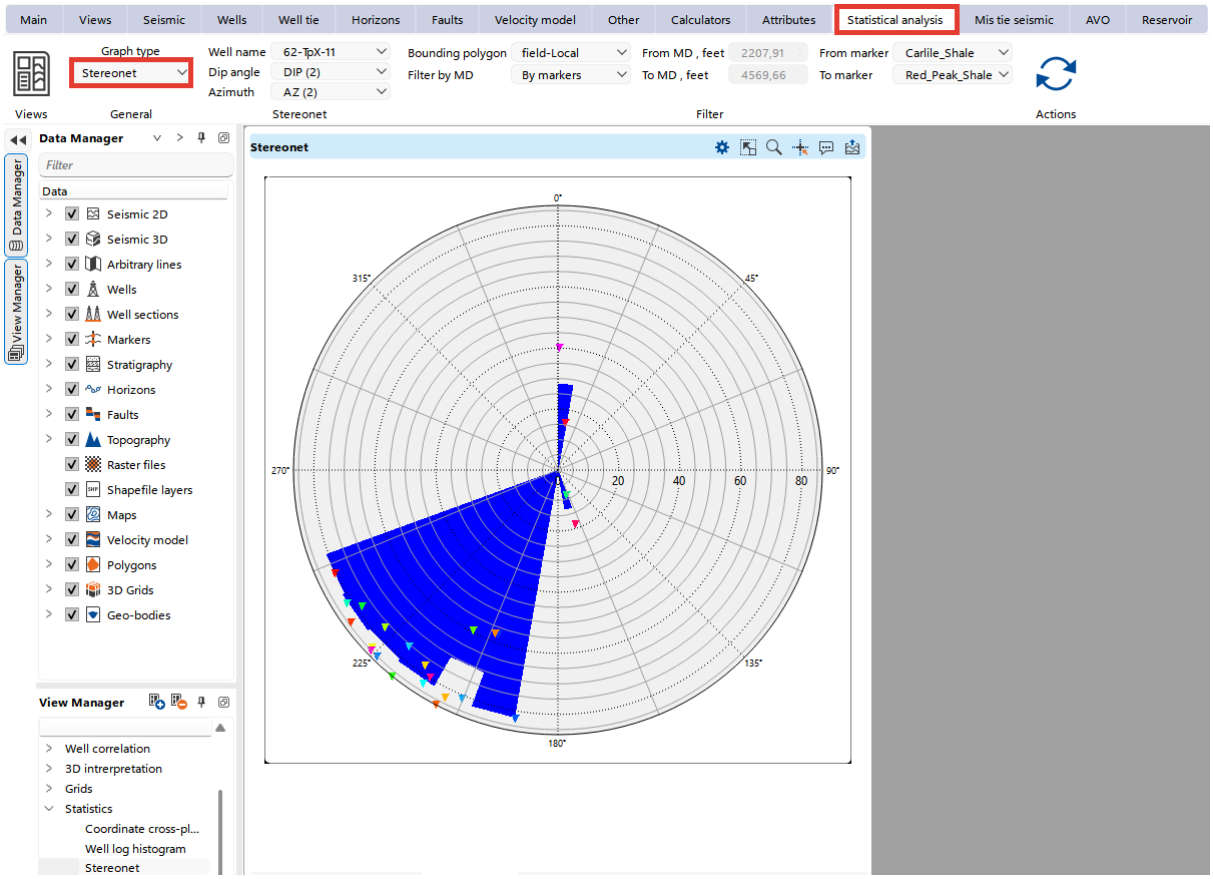
Import LAS file 407 containing **Azimuth** and **Dip** values into g-Space.

Go to the **Views bar** in the Ribbon bar. Choose Graphs - Stereonet



In the **Statistical analysis bar for Stereonet graph** select your LAS data columns for **Dip** (inclination angle) and **Azimuth** (horizontal angle).

© 2026 Geomage 2003 LTD



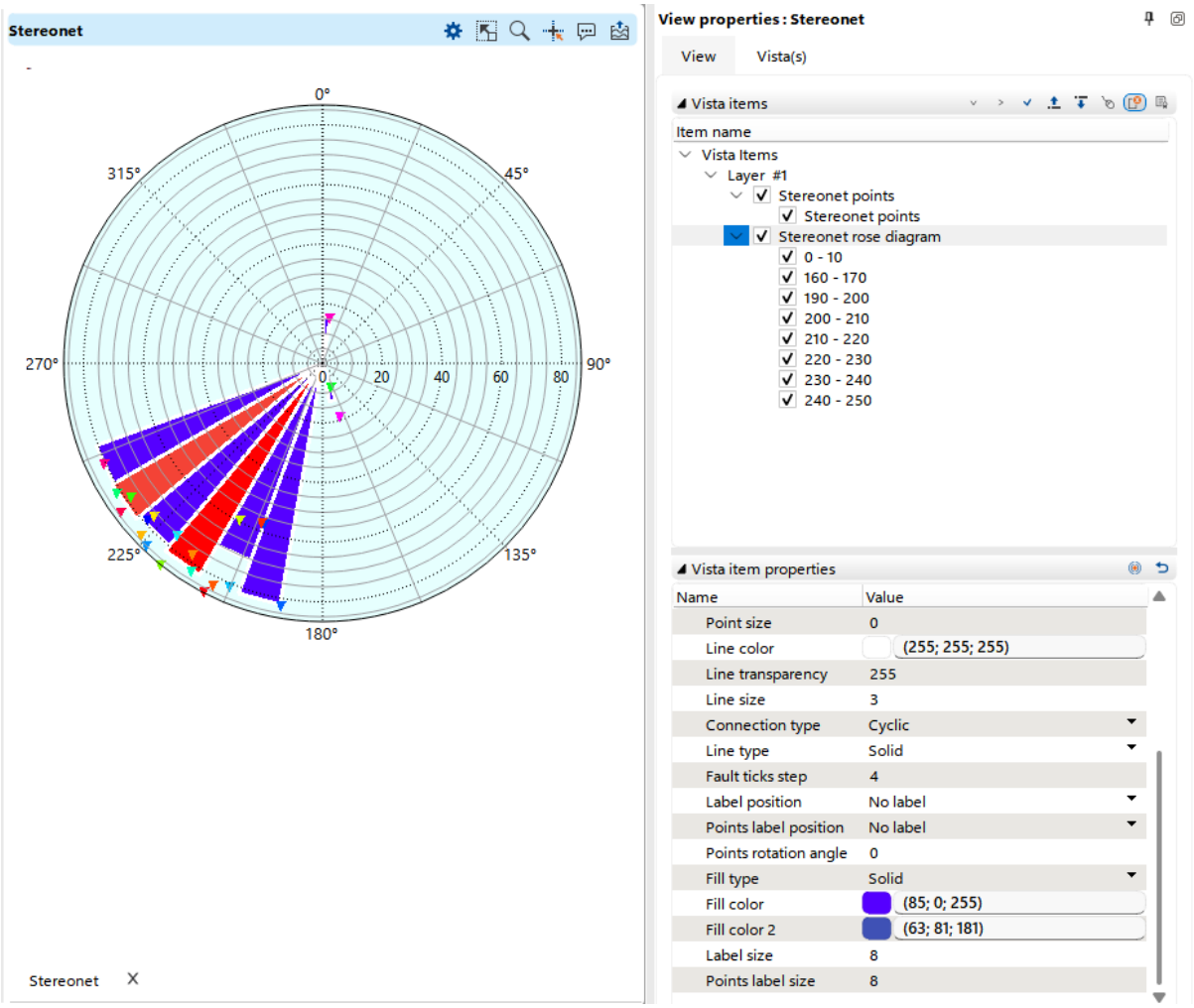
Visual Settings

Users can customize the **Stereonet** display for better interpretation:

Line and Rose Diagram Colors: Set custom colors for lines and the rose diagram to enhance clarity.

Depth-Based Color Palette: Visualize points based on **depth** using a color palette.

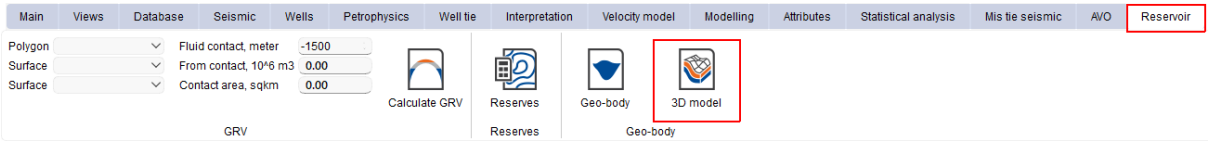
© 2026 Geomage 2003 LTD



9.13 Geological modeling

In g-Space, the geological modeling workflow allows users to create detailed structural models of subsurface geology.

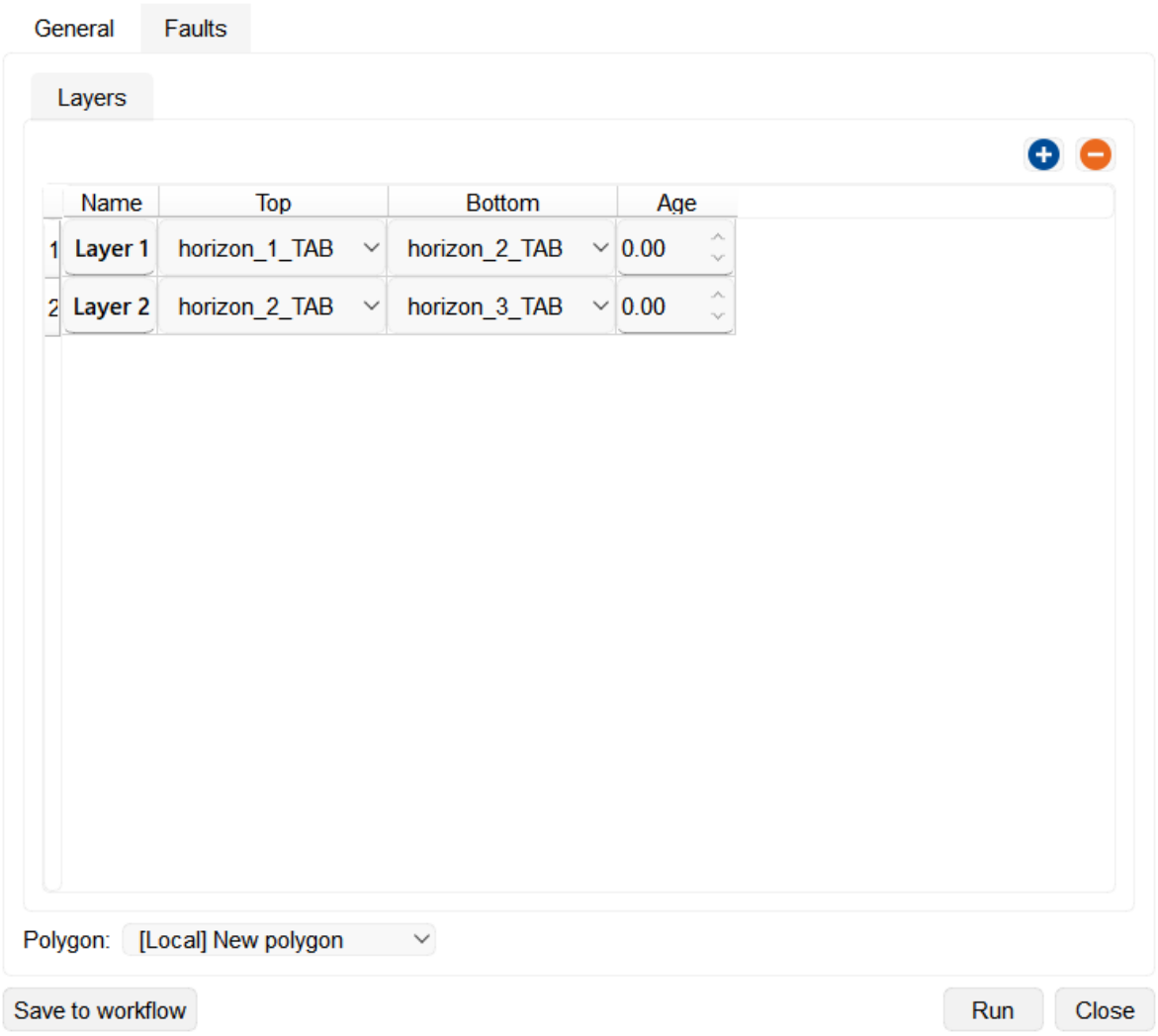
To start navigate to the Reservoir bar 122 and click Create 3D geological model.



© 2026 Geomage 2003 LTD

This process is streamlined through a user-friendly wizard that guides users through defining essential geological features such as layers and faults.

The **General** tab, currently displayed, handles layer definitions.



Use **Save to workflow** to save the current settings as a workflow task, **Run** to build the model immediately, or **Close** to exit without running. When you edit an existing workflow task, use **Save** to update it or **Cancel** to close without saving.

Layers Section allows the user to define the stratigraphic layers within the 3D geological model. Each row represents a layer in the model, with the following parameters:

- **Name**: The layer's identifier, which can be adjusted manually.
- **Top**: The geological horizon that defines the top of the layer. Users can choose a predefined depth map.

© 2026 Geomage 2003 LTD

- **Bottom**: The geological horizon that defines the bottom of the layer.
- **Age**: This column allows for defining the geological age of the layer according to **Stratigraphy** in the Data manager.

When the wizard does not have values from a previous run or workflow task, the **General** tab starts with **Layer 1**. When depth maps are available, **Top** uses the first map and **Bottom** uses the second; if only one map is available, both use that map. After you click **Run**, the next fresh launch restores the layer table and **Polygon** from that run.
Select a layer and click **"+"** to insert a layer after it, or at the end when no row is selected. The new row receives the next sequential **Layer** number, copies the preceding row's **Bottom** into its **Top**, and uses the next depth map in the list for its **Bottom**. If no map follows, **Bottom** remains the same as **Top**. You can still edit **Name**, **Top**, and **Bottom** manually; click **"-"** to remove the selected layer.

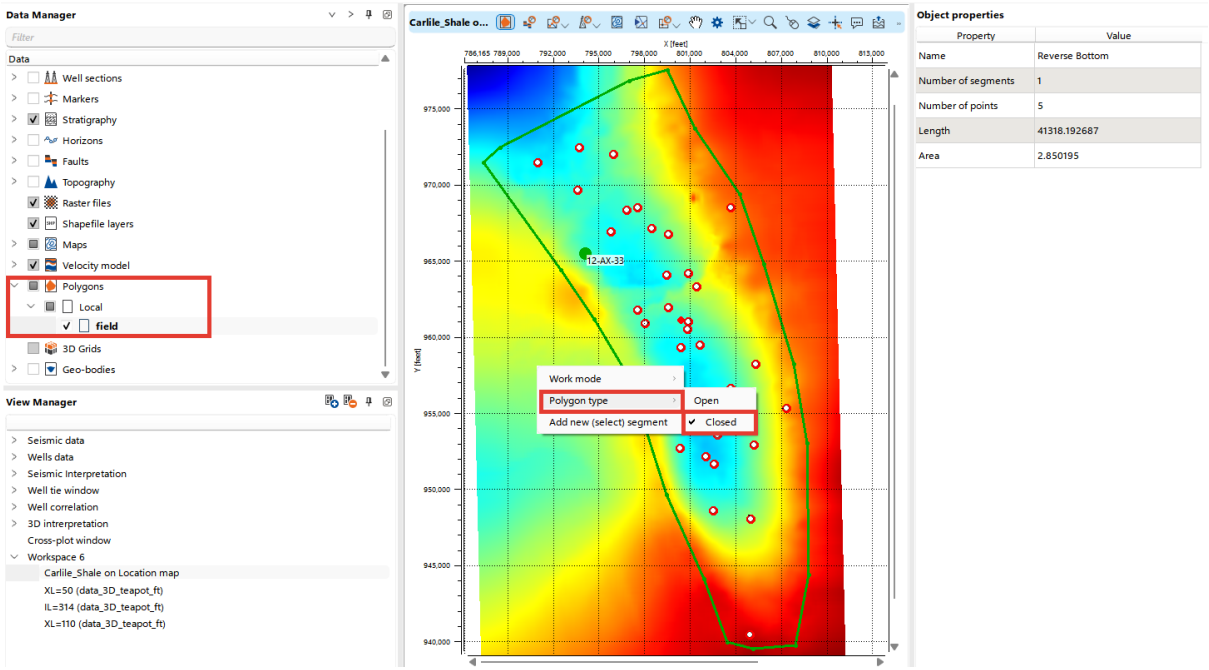
Notes

Ensure that the depth maps are derived from reliable, high-quality data, as this forms the basis of your structural model. Check for consistency between seismic-derived depth maps and well tops to avoid discrepancies in your model.

Ensure that the faults are accurately interpreted from high-quality seismic data and validated against well data or geological maps. Faults should be checked for continuity and consistency across different horizons to prevent misalignment in the model. Pay attention to fault throw and displacement, ensuring that they align with geological observations.

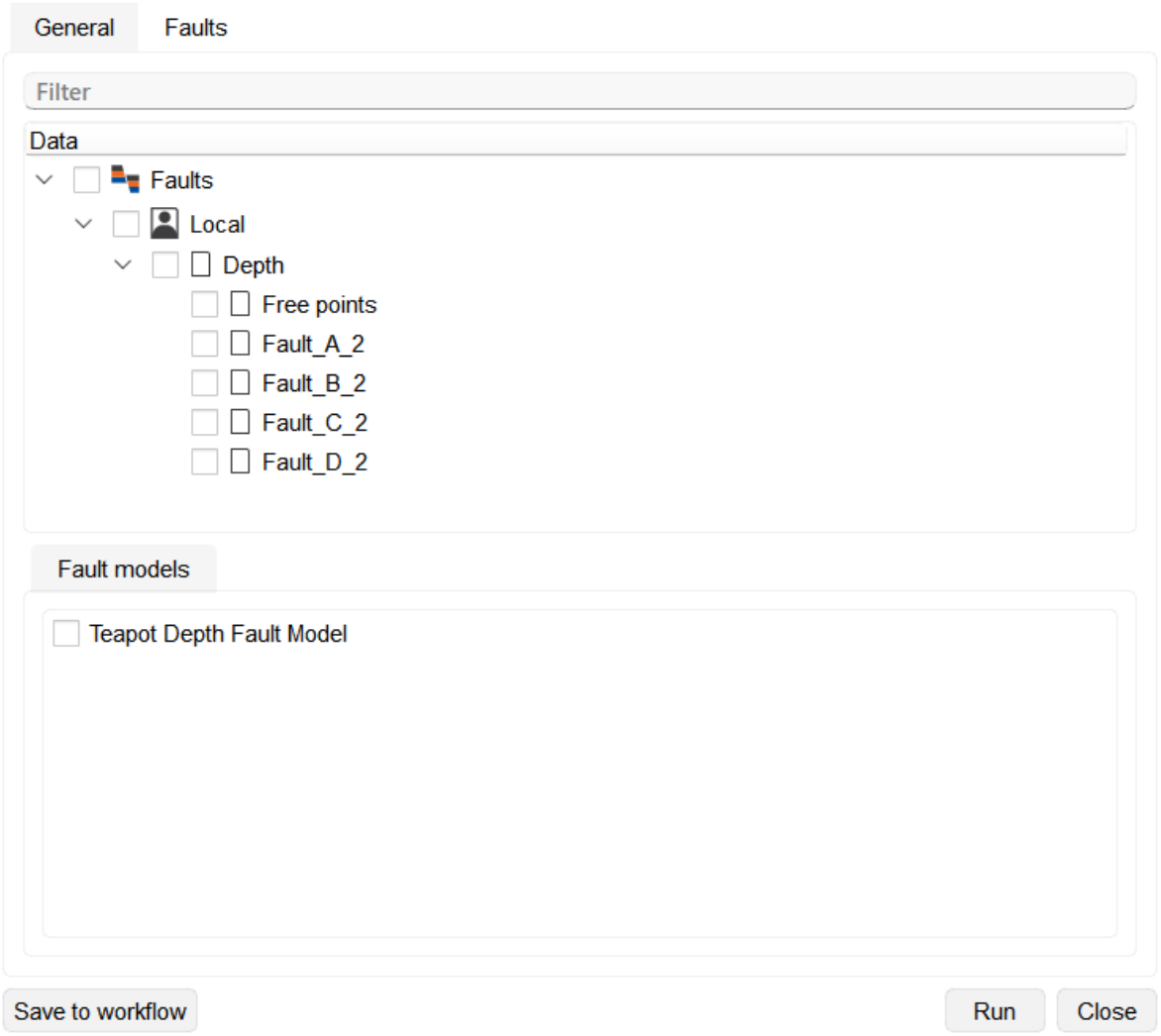
The dropdown at the bottom enables users to select a **Polygon** to define the spatial extent of the model.
When creating a polygon to define the modeling area, ensure that the polygon type is set to **Closed** by pressing **Alt + right-clicking**.

© 2026 Geomage 2003 LTD



On the **Faults** tab, select individual faults and, under **Fault models**, any depth-domain fault models that should cut the geological model. A selected fault model contributes its fitted fault surfaces together with any individually selected faults.

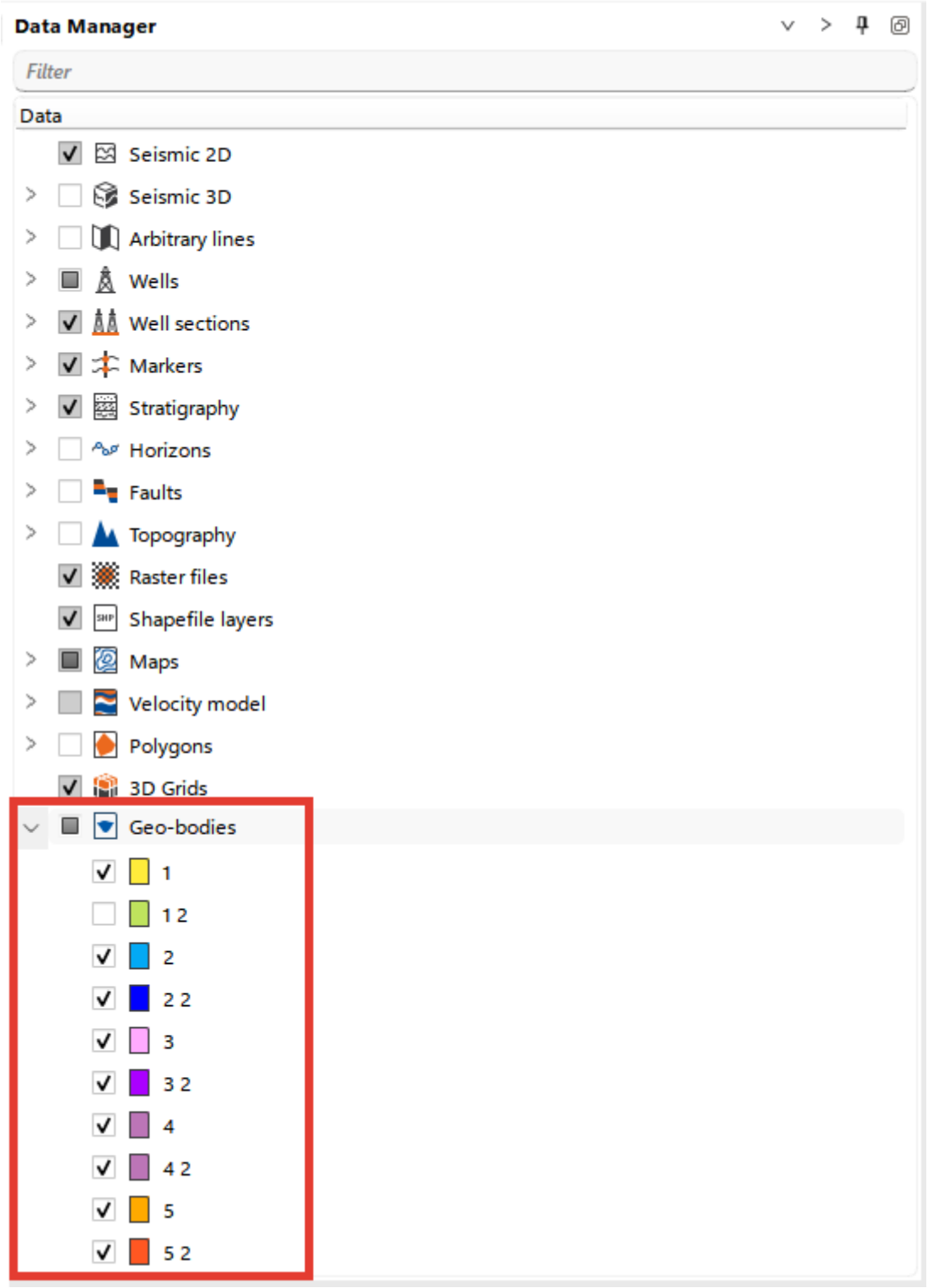
© 2026 Geomage 2003 LTD



You can see the resulting model in the Geo-bodies folder in the **Data Manager**.

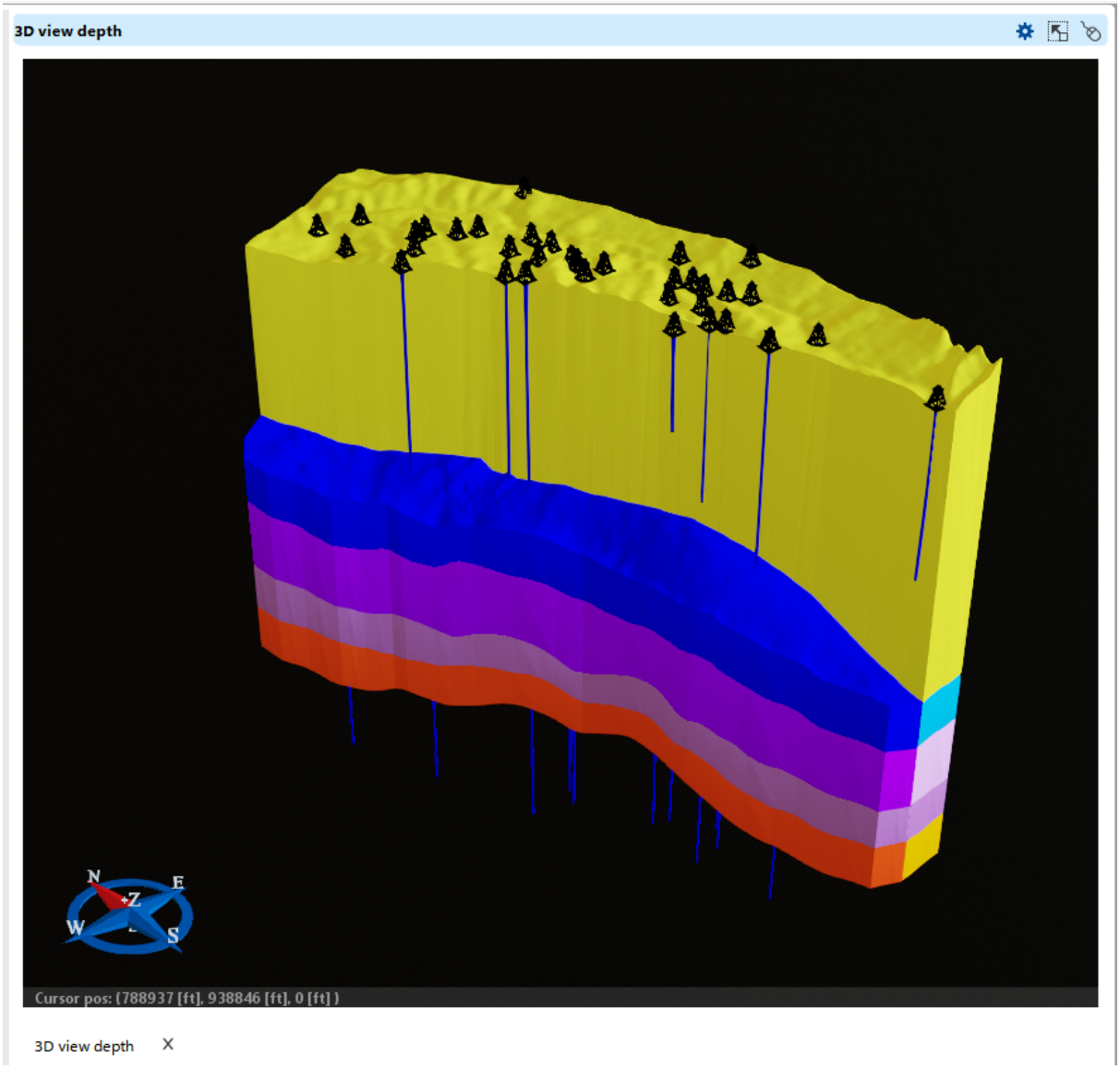
For each layer, an object will be created automatically. If the layer is divided by a fault, multiple objects with a numeric suffix will be generated, as shown in the example below, where each layer has an object with its layer number (e.g., 1), along with additional objects for faulted segments (e.g., 1.2). To rename any object, simply right-click on it and select **Rename** from the menu.

© 2026 Geomage 2003 LTD



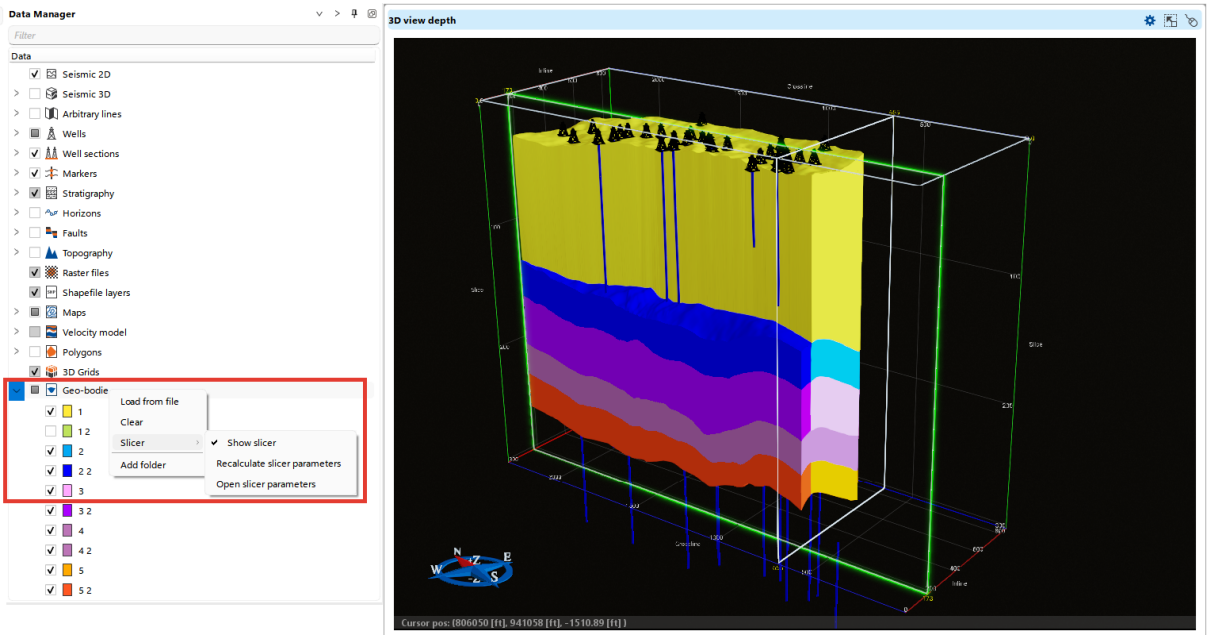
You can visualize the model in **3D view**, as well as in sections. Each object can be visualized individually via the **Data Manager**.

© 2026 Geomage 2003 LTD



The **Slicer** mode, accessible by right-clicking on the **Geo-bodies** folder in the **Data Manager**, allows you to interactively view geomodel sections in 3D. The active slicing plane is highlighted in light green, as shown in the example figure, where the plane is oriented along the field.

© 2026 Geomage 2003 LTD



9.14 3D grid creating

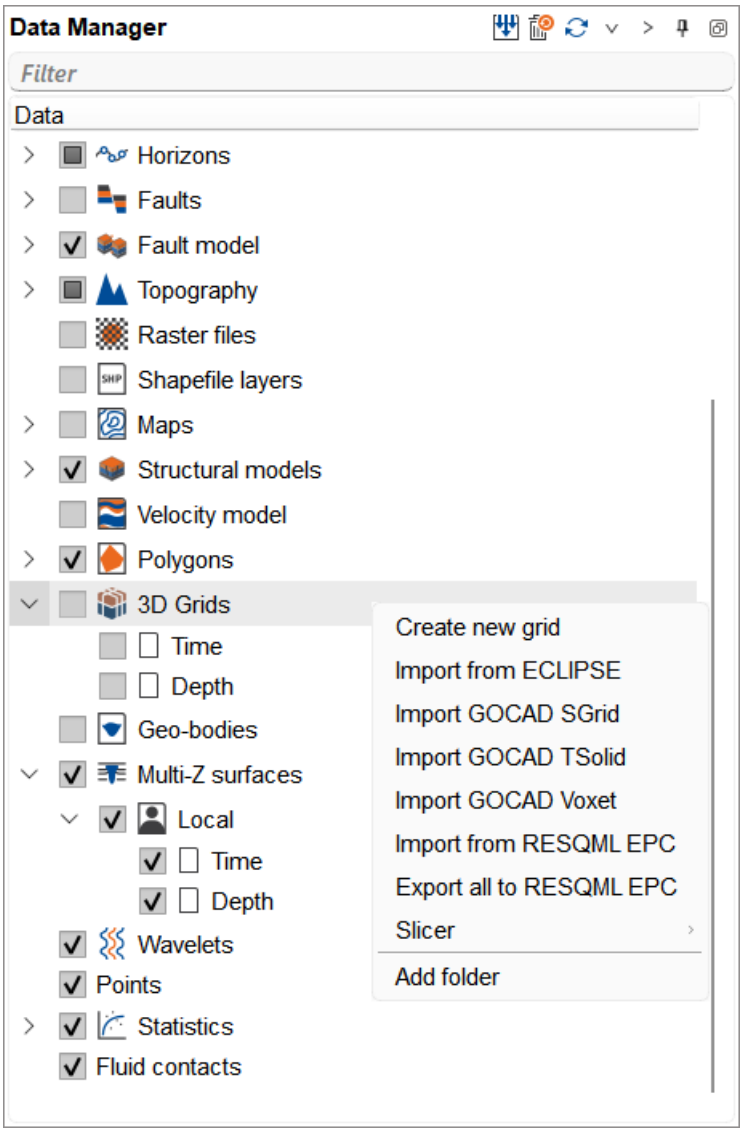
3D grids are essential tools used to organize and visualize subsurface data in a structured format. The process typically involves the interpolation of scattered data points (e.g., well tops or seismic picks) across a predefined grid.

Grids are built using interpolation techniques that estimate unknown values between data points. The grid can vary in resolution, based on the interpolation steps defined by the user, allowing flexibility in adapting to different geological complexities.

To begin the 3D gridding process in g-Space navigate to the **Data Manager**, find the 3D Grids section. This is where grid data will be organized and stored.

To create a grid, right-click the **3D Grids** folder and select **Create new grid**. In **Select grid type**, choose **S-Grid** or **Tetrahedral** and click **OK**.

© 2026 Geomage 2003 LTD



To create a tetrahedral grid, first run a structural model with at least two horizon result maps. In **Create Tetrahedral Grid**, enter the **Grid name**, select the **Structural model**, and choose a **Layering type**. For **Proportional** layering, set the **Layers per zone**; for **Parallel to Top** or **Parallel to Bottom**, set the **Layer thickness**. Set the horizontal interpolation steps, then click **OK** to create the grid.

For an S-Grid, enter the **Grid name**, choose an optional **Polygon** and whether to **Extrapolate in the polygon**, set the horizontal interpolation steps and **Margin**, and select the **Input type**. The **Faults** group defines whether faults are taken into account when the grid surfaces are built.

- **Polygon**: If available, you can select a polygon to limit the grid to a particular area. Choose "Not required" if the grid is intended to cover the full extent of the data.

© 2026 Geomage 2003 LTD

- **Interpolation Step X [ft/m]** and **Interpolation Step Y [ft/m]** These define the resolution of the interpolated data. Smaller steps result in higher grid resolution, capturing finer geological details but increasing computational demand. Larger steps provide a more generalized view but are faster to compute.

Notes

Geological Context: In regions where fine stratigraphic details or thin reservoirs are critical (e.g., offshore channels, thin carbonate layers), smaller steps (10-20 ft/1-5 m) may be appropriate. In contrast, for larger, homogenous geological structures (e.g., broad anticlines or synclines), larger steps (50-100 ft/20-50 m) may suffice.

- **Margin [ft/m]**: This parameter sets an additional boundary around the grid. A larger margin might be useful in areas where you expect the geology to extend beyond known data points.
-

In **Faults**, **Mode** defines how faults are used when the grid surfaces are built:

- **Ignore faults**: the default. The grid surfaces are built without using faults.
- **Use selected faults**: shows the **Select faults** list, where you select the check box of every fault to apply. If no fault is selected, the grid is built without using faults.
- **Use fault model**: shows the **Fault model** list, where you select one existing fault model.

Select faults and **Fault model** offer only the objects that match the domain of the selected **Input type**: time faults and time fault models for **Time horizons** and **Time maps**, depth faults and depth fault models for **Depth horizons** and **Depth maps**. Changing the **Input type** rebuilds both lists and clears any selection that is not available in the new domain.
For both fault modes, **Clear radius [ft/m]** sets how far from a fault trace the grid values are recalculated, so that the surface is no longer interpolated across the fault. Faults classified as **Normal** also leave a gap along their trace; fault types are assigned in the Faults 158 branch of the Data Manager.

In **Create grid**, click **Run** to create the S-Grid. Click **Save to workflow** to store the setup as a workflow task, or **Close** to dismiss the window. Run and Save to workflow leave the window open.

To review or change the created S-Grid's physical extent, right-click the grid and select **Settings**. The **Grid parameters** window controls the grid's position, rotation, spacing, and size. It also provides the same **Faults** group as the creation window, so fault handling can be changed after the grid exists. Here the fault and fault model lists follow the grid's own domain, and applying a change rebuilds the grid surfaces with the current fault settings.

- **Start X [ft/m]** and **Start Y [ft/m]**: These values define the starting coordinates of the grid in the X and Y directions.
- **Step X [ft/m]** and **Step Y [ft/m]**: These values determine the spacing between the grid nodes along the X and Y axes. Smaller grid steps result in a finer grid, while larger grid steps produce a coarser, more general representation.

Notes

© 2026 Geomage 2003 LTD

Geological Context: For detailed geological structures, use smaller grid steps (e.g., 10-20 ft/1-5 m). Larger grid steps (e.g., 50-100 ft/20-50 m) can be used for large-scale structural features.

- **Angle [degree]**: This defines the rotation of the grid relative to the coordinate system.

Notes

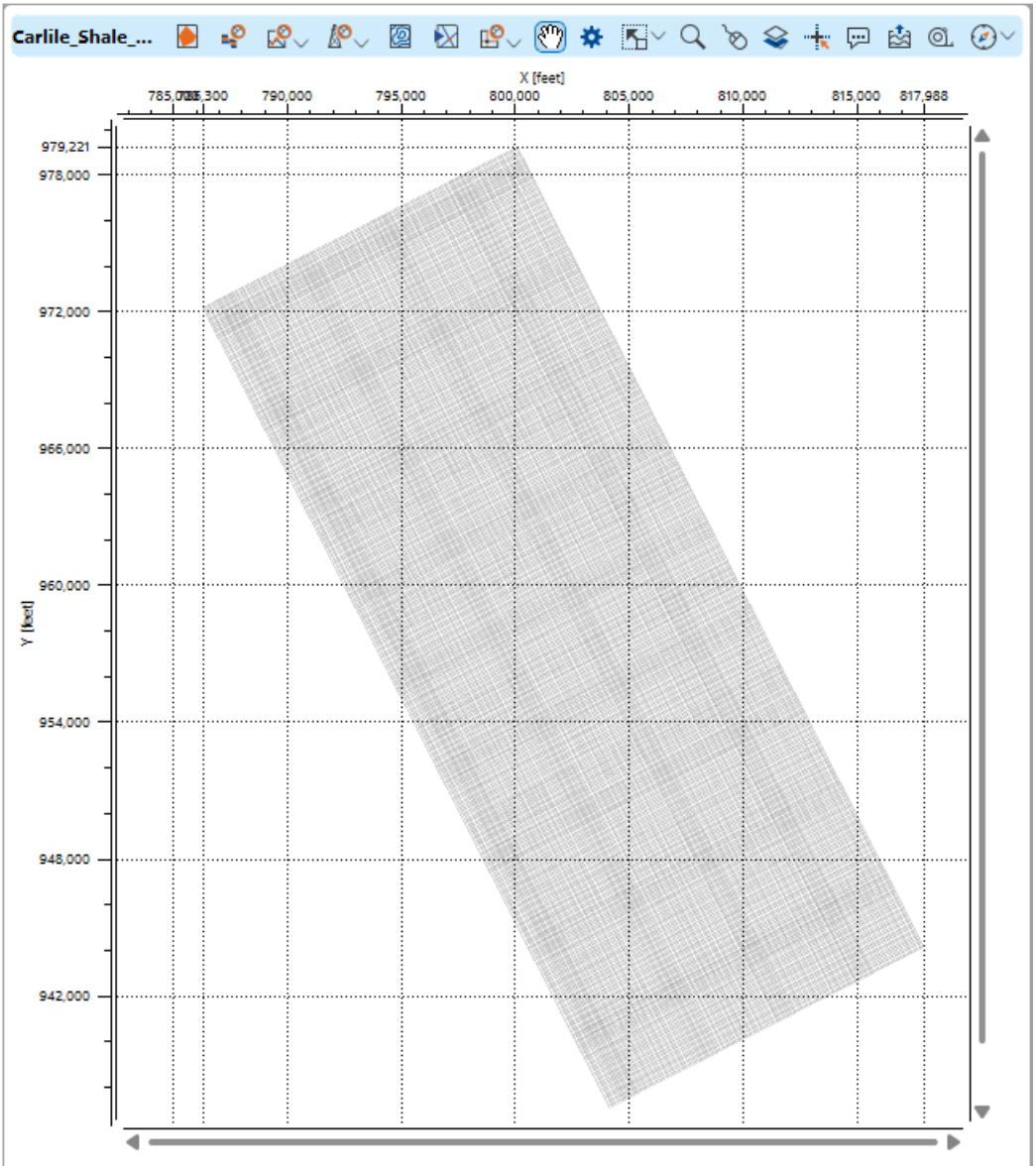
Grid rotation is essential for aligning the grid with key features to improve accuracy and efficiency:

1. ***Tectonic Structure Alignment**: Rotate the grid to match elongated tectonic features, minimizing the number of cells inside the reservoir contour while preserving horizontal resolution.*
2. ***Well Patterns**: Aligning the grid with injection and production wells ensures more direct fluid flow paths, reducing filtration time and increasing efficiency.*
3. ***Fault Orientation**: Rotating the grid along fault trends prevents grid distortion near faults, maintaining smooth geological transitions.*

- **Length X [ft/m]** and **Length Y [ft/m]**: These define the total dimensions of the grid, controlling how far the grid will extend in the X and Y directions. Adjust these based on the size of your study area.

In **Grid parameters**, click **Run** to apply the changes, **Save to workflow** to preserve the setup, or **Close** to dismiss the window. Run and Save to workflow leave the window open.

© 2026 Geomage 2003 LTD



Right-click a grid in the **3D Grids** section of the **Data Manager** to work with it. Available actions depend on the grid type. The options below describe an S-Grid. Tetrahedral grids omit the S-Grid-only **Edit zone**, **Update**, **Export to ECLIPSE**, **QC**, and **Upscale seismic volume** actions. Their menu instead provides **Export to VTK**, **Export to GRDECL**, **Export to GOCAD TSolid**, **Convert to Geobodies**, and **Mesh optimization**.

Settings: Opens **Grid parameters**, where you can change Start X, Start Y, Step X, Step Y, Angle, Length X, Length Y, and the fault settings.

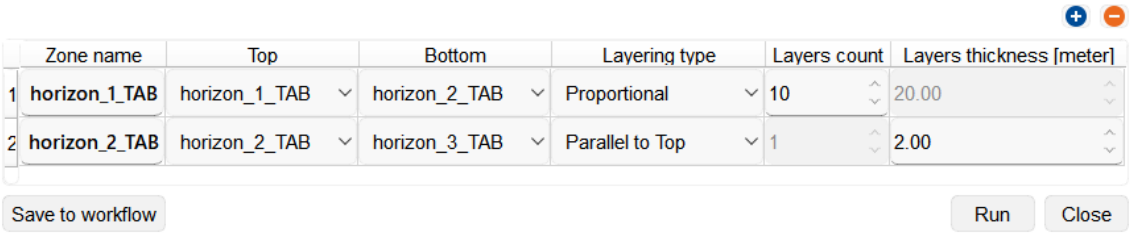
Edit zone: Opens **Manage grid**. Add or remove zones with **+** and **-**, select the **Top** and **Bottom**, choose a **Layering type**, and set **Layers count** or **Layers thickness** as applicable. Click **Run** to apply the changes, **Save to workflow** to preserve the setup, or **Close** to dismiss the window. Run and Save to workflow leave the window open.

© 2026 Geomage 2003 LTD

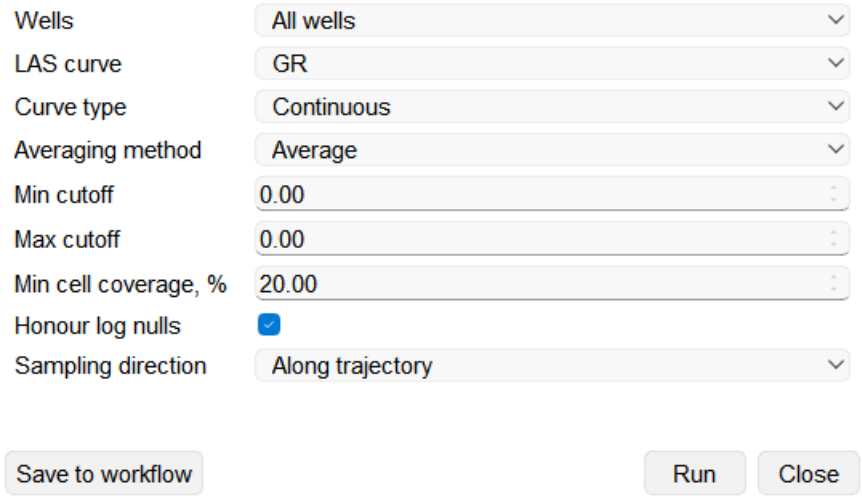
Notes

The number of layers in a zone, is influenced by several factors:

1. ***Geological Heterogeneity:*** *The vertical variability in properties, like the average thickness of reservoir layers, dictates layer thickness. For instance, if the average layer thickness is 1 meter, grid cells larger than 50 cm shouldn't be used to capture details effectively. This is assessed using histograms of layer thickness.*
2. ***Well Data Sampling Interval:*** *Typically, well data is sampled at intervals of 0.2 to 0.4 meters, which affects layer thickness.*
3. ***Total Thickness:*** *The entire thickness of the interval to be subdivided impacts the number of vertical layers required.*
4. ***Computer Limitations:*** *The computational power available may restrict the grid resolution that can be practically implemented.*



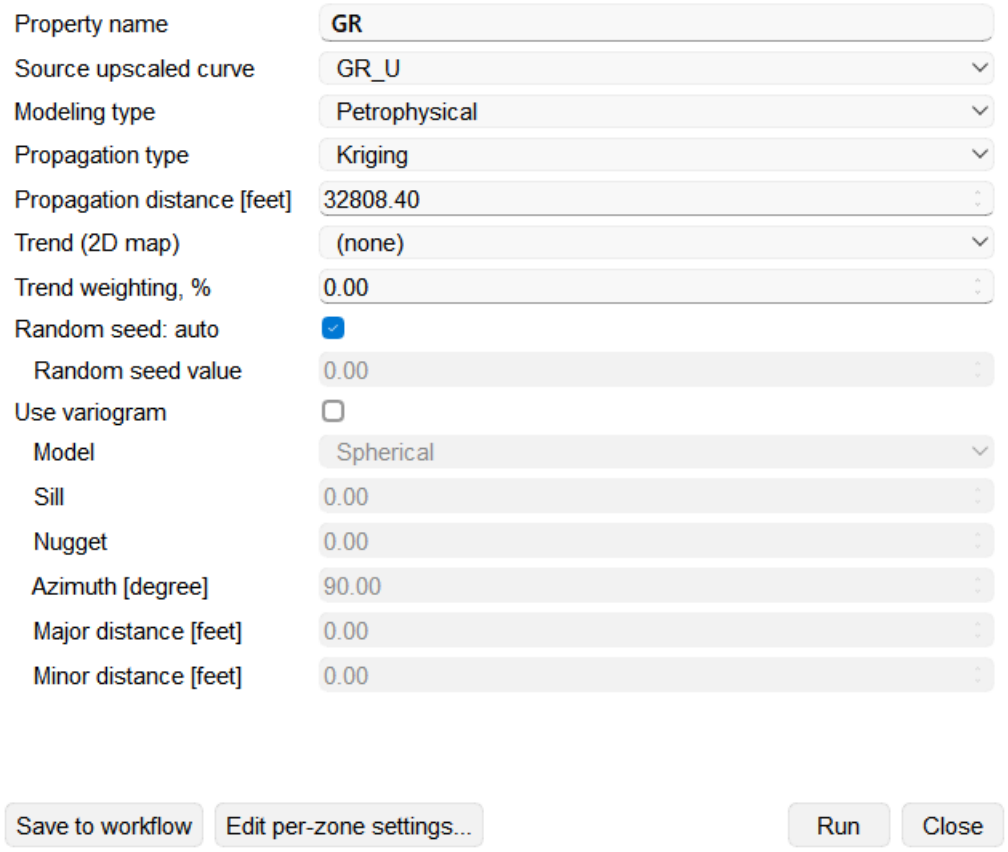
Upscale Log Data: Creates cell values from a selected LAS curve. Choose the wells, curve and curve type, averaging method, cutoffs, minimum cell coverage, null handling, and sampling direction. Click **Run** to store the result under **Upscaled cells**, or **Save to workflow** to preserve the setup for reuse.



Propagate Upscaled Properties: Creates a grid property from an item under **Upscaled cells**. Select the source upscaled curve and the modeling and propagation settings. Use

© 2026 Geomage 2003 LTD

Edit per-zone settings... when different zones need different parameters, then click **Run**. **Save to workflow** preserves the setup for reuse.



Update: Refreshes or recalculates the grid, applying any recent changes in data or grid parameters.

Export to ECLIPSE: Allows the user to export the grid to the **ECLIPSE** format, which is widely used for reservoir simulation and modeling. This function is critical for taking the grid data from g-Space into reservoir simulation software.

Rename: Renames the selected grid in the Data Manager for organizational purposes.

Remove: Deletes the selected grid from the project.

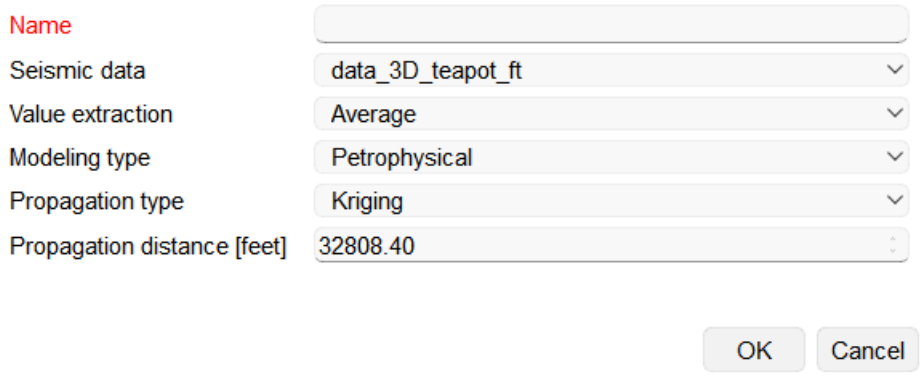
Show grid report: Shows the report of the selected grid in the **Properties table** of the Table view[285]. For an S-Grid, the report displays general parameters of the grid, such as dimensions, resolution, and volume, and provides detailed information for each zone within the grid. For a tetrahedral grid, it holds only a single Entire grid row with the cell count and volume, followed by one statistics block with the rows Min, Max, Average, Median, Rms, and Most common and one column per grid property; there are no Step X and Step Y values, no Top, Bottom, or Layers columns, and no per-zone tables. The data from the grid report can be copied directly or saved as a *.CSV* file

© 2026 Geomage 2003 LTD

Properties table: 3D grid - Grid 1

Grid 1						
	Step X:	Step Y:				
	200.000 m	200.000 m				
		Top	Bottom	Layers	Cells	Volume
	horizon_1_TAB	horizon_1_TAB	horizon_2_TAB	10	9950	11200695296.000
	horizon_2_TAB	horizon_2_TAB	horizon_3_TAB	183	182085	12845818880.000
Entire grid	Top, m	Bottom, m	Vertical...	LITHO	Poro	
Min	208.248	776.683	535.228	2.000	0.000	
Max	464.283	1117.322	684.159	3.000	0.295	
Average	344.392	948.576	604.184	2.787	0.239	
Median	356.332	960.042	599.417	3.000	0.248	
Rms	348.435	951.931	605.222	2.817	0.247	
Most common	356.359	926.579	570.220	3.000	0.000	

Upscale seismic volume: For an S-Grid, samples a selected seismic volume into the grid. Enter a **Name**, select **Seismic data** and **Value extraction**, set the modeling and propagation parameters, and click **OK**.



Advanced Options and Applications

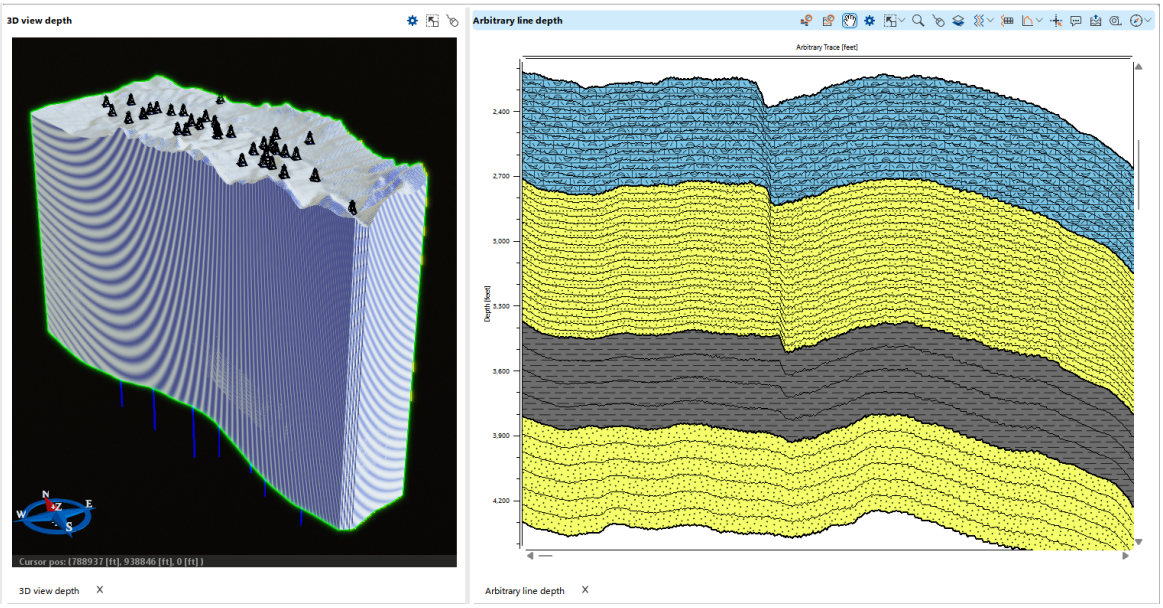
Polygon Clipping: If you are working in a specific region of interest, using a polygon to limit the grid's extent can reduce computation time and ensure focus on relevant data.

Time vs. Depth Grids: Grids can be created in both time and depth domains. A grid is built from time or depth horizons and maps and keeps its native domain — a grid built from

© 2026 Geomage 2003 LTD

time inputs stays in time, with no depth conversion applied. Depth grids incorporate well data and velocity models for more accurate geological interpretations. Well logs (LAS) can be upscaled onto a time-domain grid provided the wells carry a checkshot (time-depth relationship), which positions the log samples in time.

3D grid can be visualized in 3D depth view as well as in section view. Visualization settings can be configured in the Visual Settings 206, in the example below, lithological fill type is selected for the **Grid zones**.



9.15 Mesh optimization

Mesh optimization improves the geometric quality of an existing triangulated mesh. Meshes that were edited by hand or assembled from several datasets often become unnecessarily dense, contain badly shaped elements, or are left with gaps at surface contacts. The tool can even out such a mesh, reduce its element count, rebuild it with a more uniform element size, and close open holes.
The operation always produces a **new** object in the Data Manager 124 — the source surface or grid is never modified, so you can compare the two and keep the one you prefer.

Supported objects

© 2026 Geomage 2003 LTD

- Triangulated Multi-Z surfaces 178. All four operations are available. A surface that still consists of sticks only is not triangulated yet — build its mesh with **Recreate surface** first.
- Tetrahedral 3D grids 169. Only **Smoothing** can be run on them, but **Quality metrics** is reported for them as well.

Only these two object types appear in the dialog. Structured grids (S-Grids) are not mesh-optimization inputs.

Opening the dialog

- In the **Data Manager**, right-click a surface under **Multi-Z Surfaces** and choose **Mesh optimization**. The entry stays visible but is unavailable for a surface that has no triangulated mesh.
- In the **Data Manager**, right-click a tetrahedral grid under **3D Grids** and choose **Mesh optimization**. The entry is not offered for structured grids.
- Or run **Mesh optimization** from the Actions 214 panel and pick the object inside the dialog.

The object you right-clicked is preselected in the dialog. If the project contains neither a triangulated surface nor a tetrahedral grid there is nothing to optimize, and the dialog does not open.

Input object

Input object is shared by both tabs and lists every mesh in the project that can be optimized. Tetrahedral grids are marked with **(TetGrid)** after the name so they can be told apart from surfaces of the same name. Changing the selection updates the available operations and the reported metrics.

Optimize

Select the **Operation** and set its value in the row below. The row label and the editor change with the selected operation.

- **Smoothing** — set **Smoothing %** between 0 and 100 (default 50). Vertices are moved to even out the mesh while the overall shape is preserved. On a tetrahedral grid only the outer boundary is smoothed; interior nodes, the cell count and the connectivity stay unchanged.
- **Decimation** — set **Target ratio %** between 10 and 100 (default 50). The mesh is simplified until roughly that share of the original triangle count is left. Boundary edges are preserved, so the outline of the surface does not shrink.
- **Re-mesh** — set **Target edge length**. The mesh is rebuilt with triangles of a more uniform size. Leave the value at **(median)** to use the median edge length of the input mesh. The outline of the patch is preserved.

© 2026 Geomage 2003 LTD

- **Topology repair** — set **Max hole edges (0 = unlimited)**. Open border loops are triangulated and smoothed into the surrounding mesh, which is how you close gaps and work towards a watertight model. A loop with more edges than the limit is skipped; leave the value at **(unlimited)** to fill every hole that is found.

Press **Run** to create the optimized object, or **Cancel** to close the dialog without changes. **Run** is unavailable when the selected combination cannot be applied: on a tetrahedral grid every operation except **Smoothing** is blocked, because it would change the cell count or the connectivity of the grid.

Quality metrics

The **Quality metrics** tab reports read-only measurements of the object selected in **Input object**. Use it to decide which operation is needed, and afterwards to compare the result with its source.

- Element and vertex counts — triangles for a surface, tetrahedra for a tetrahedral grid.
- **watertight** — **yes** when the mesh is closed, **no** when it still has open borders.
- **non-manifold vertices** — the number of vertices where the mesh is topologically inconsistent. They are reported for information only and are not corrected automatically.
- **Edge length** — minimum, maximum, mean and standard deviation. A wide spread points to an uneven mesh that **Re-mesh** can regularize.
- **Dihedral angle (deg)** — minimum, maximum, mean and standard deviation of the angles between adjacent faces. Very small or very large angles indicate distorted elements.
- **Aspect ratio histogram (inradius / circumradius)** — the number of elements in ten bins from 0.0-0.1 to 0.9-1.0. A perfectly equilateral element scores 1.0, so a distribution weighted towards the right-hand bins means better-shaped elements.

When the measurements cannot be computed for the selected object, the tab reports that metrics are unavailable for this input instead of showing values.

Results

- A new object appears next to the source in the Data Manager. Its name is the source name plus a suffix naming the operation — **(smoothed)**, **(decimated)**, **(remeshed)** or **(repaired)**. A counter is added if that name is already in use.
- An optimized Multi-Z surface is created under the same **Time** or **Depth** domain node as its source and keeps its colour.
- An optimized tetrahedral grid is created under **3D Grids** as a complete copy of the source grid, including its properties and layers.
- Nothing is created when the settings would leave the mesh unchanged — for example **Smoothing** at 0 %, **Decimation** at 100 %, or **Topology repair** on a mesh that has no holes within the chosen limit.

© 2026 Geomage 2003 LTD

- Select the new object in **Input object** and open **Quality metrics** to confirm the improvement before you continue modelling with it.

Recommendations

- Read **Quality metrics** before you choose an operation: the watertight flag and the edge-length spread usually tell you whether the mesh needs repairing, regularizing or only smoothing.
- Close the gaps first. Run **Topology repair** while the mesh reports **watertight: no**, then smooth or re-mesh the repaired result.
- Work in moderate steps and check the metrics after each run, rather than applying one aggressive setting.
- Use **Decimation** when the mesh is simply too heavy, and **Re-mesh** when the element sizes are uneven.

See Also

- Multi-Z Surfaces 178
- 3D Grids 169
- 3D grid creating 765
- 3D Grid Quality Control 172
- Data manager 124

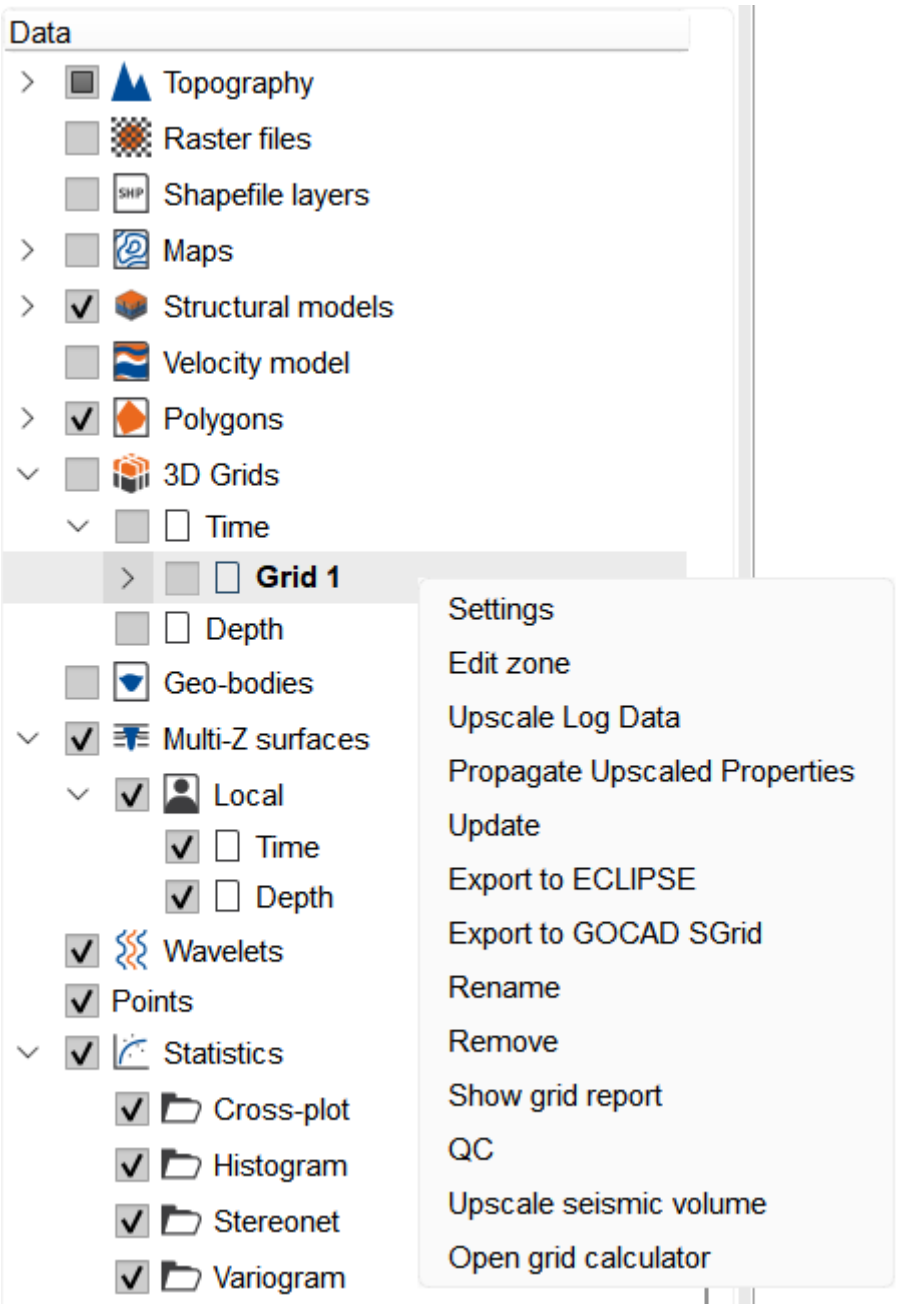
9.16 Properties modeling

In g-Space, **Properties modeling** involves the integration and calculation of well log data and other geological properties to populate 3D grids 765 with meaningful information, such as porosity, permeability, and fluid saturation.

To start, navigate to the Data Manager, Select the desired grid (e.g., "Grid 1") by clicking on it. This action highlights the grid and makes it the active selection.

Right-click on the selected grid to open the context menu. This menu contains various options related to the selected grid, allowing for efficient data management.

© 2026 Geomage 2003 LTD



Upscale Log Data blocks (upscales) the selected well log curve onto every 3D grid cell that a wellbore trajectory crosses, averaging the log samples that fall inside each cell. Only cells crossed by a well receive a value; all other cells stay empty, so you can QC the conditioning data before property modelling. The **LOG upscaling parameters** window will appear, where you choose the wells, the LAS curve and how the samples are averaged.

© 2026 Geomage 2003 LTD

1. **LAS Curve Selection:** The user can select the type of curve from the dropdown, e.g., DT (Sonic), GR (Gamma Ray), etc.

2. **Averaging method:** dropdown offers multiple statistical methods to extract well log data into the grid:
   - **Average**: Computes the length-weighted average of the log samples that fall in the cell. Each sample is weighted by the length of well trajectory it represents inside the cell (the sum of value × segment length divided by the sum of segment lengths), so uneven sample spacing and trajectory angle do not bias the cell value.
   - **RMS**: Extracts the Root Mean Square value, providing a weighted average of the data, emphasizing higher values.
   - **Minimum**: Selects the minimum value of the LAS curve in the given interval.
   - **Maximum**: Extracts the highest value of the LAS curve in the interval.
   - **Most common**: Identifies the most frequent or significant value within the LAS data.
   - **Median**: Extracts the median, giving the middle value of the LAS curve data, minimizing the impact of extreme values.

3. **Curve type:** select **Continuous** or **Discrete**. The type is auto-detected from the LAS curve and can be overridden. It filters the Averaging method above: continuous curves offer Average, RMS, Minimum, Maximum and Median; discrete curves offer **Most common** and **Percentage**. Choosing Percentage reveals a **Facies code** field; the cell value is then the fraction of the in-cell trajectory where the curve equals that code (0–1).

4. **Min cutoff / Max cutoff:** samples below **Min cutoff** or above **Max cutoff** are discarded before averaging; a sample exactly on a bound is kept. Both fields open on the data range of the selected **LAS curve** — the lowest and highest valid values it holds across the wells currently selected, ignoring null and invalid samples — rounded outward so an automatic cutoff never clips a real sample, and shown with two to six decimals according to the size of those values, so a small-range curve is displayed more precisely. They are recalculated the same way whenever **Wells**, **Folder**, **Filter**, **Selected wells** or **LAS curve** changes, replacing whatever the fields held, but they stay editable and keep a value you type until one of those selections changes again. A stored pair whose **Min cutoff** is lower than its **Max cutoff**, for example from a saved workflow step or an earlier run, reopens unchanged. If the selected curve has no valid values in the selected wells, both fields are set to 0.00 and the **Min cutoff** and **Max cutoff** labels turn red, with the tooltip "Selected log has no valid values in the selected wells". Equal bounds apply no cutoff.

© 2026 Geomage 2003 LTD

5. **Min cell coverage, % and Honour log nulls: Min cell coverage, %** skips cells the trajectory only clips (in-cell length below the given percentage), avoiding spurious single-sample values on grazing wells. **Honour log nulls** respects the LAS NULL value so null samples never enter the average.

6. **Wells:** choose which wells contribute — **All wells**, a **Well folder**, a **Well filter**, or **Individual wells** selected by name. **Well folder** and **Well filter** are offered only when the project contains at least one well folder or well filter. Choosing either one shows a **Folder** or **Filter** list of the well folders and well filters that exist in the project, and only the wells in the folder or filter you pick are upscaled; the wizard reopens on your last pick. Choosing **Individual wells** shows a **Selected wells** field holding a comma-separated list of well names; it opens on your previous selection, or on every well that carries the selected LAS curve when there is none, and wells that share a name are told apart by the UWI shown after the name.

7. **Sampling direction: Along trajectory** (default) places each sample at its true position on the wellbore from the deviation survey, so cell membership follows the 3D path; **TVD** treats the well as a straight vertical line under the surface location, useful when the deviation survey is missing or unreliable.

On **Run**, the upscaled curve is written to a new **Upscaled cells** folder under the grid, named **<curve>_U** (for example RHOB_U). Cells crossed by a well hold the averaged value while all other cells stay empty. A per-well report is then shown, listing for each well the cells touched, samples used, and samples discarded by cutoff or null. Use **Save to workflow** to store the settings as a reusable workflow step.
Each upscaled curve under **Upscaled cells** carries its own check box in the Data Manager. Select it to show, in the 3D view, only the cells that actually took a value from the log, drawn as grid cells along the well path; each curve is switched on and off independently. Curves upscaled with the **Most common** method are colored by lithology category, and the **Visual Settings** panel then shows a read-only **Categories** list of the codes present and the color each one gets; all other upscaled curves are colored by value. The cells follow the grid's domain, so they appear in the 3D view for that domain, and both S-grids and tetrahedral grids are supported.

Propagate Upscaled Properties

Once the LAS data is assigned and processed, this option allows the user to save the well log data (LAS) as a property within the grid. This property can then be used for subsequent geological and reservoir calculations, modeling, and further analysis.
Name the property, then select the **Modeling type** — either Petrophysical or Facies Modeling.

g-Space supports two types of Modeling properties:

© 2026 Geomage 2003 LTD

Petrophysical: These are scalar properties like porosity, permeability, and saturation that vary smoothly across the reservoir. Choose this type for continuous curves.

Facies: These include categorical data such as lithology, facies codes, or rock types, typically constrained by facies boundaries. Choose this type for discrete curves.

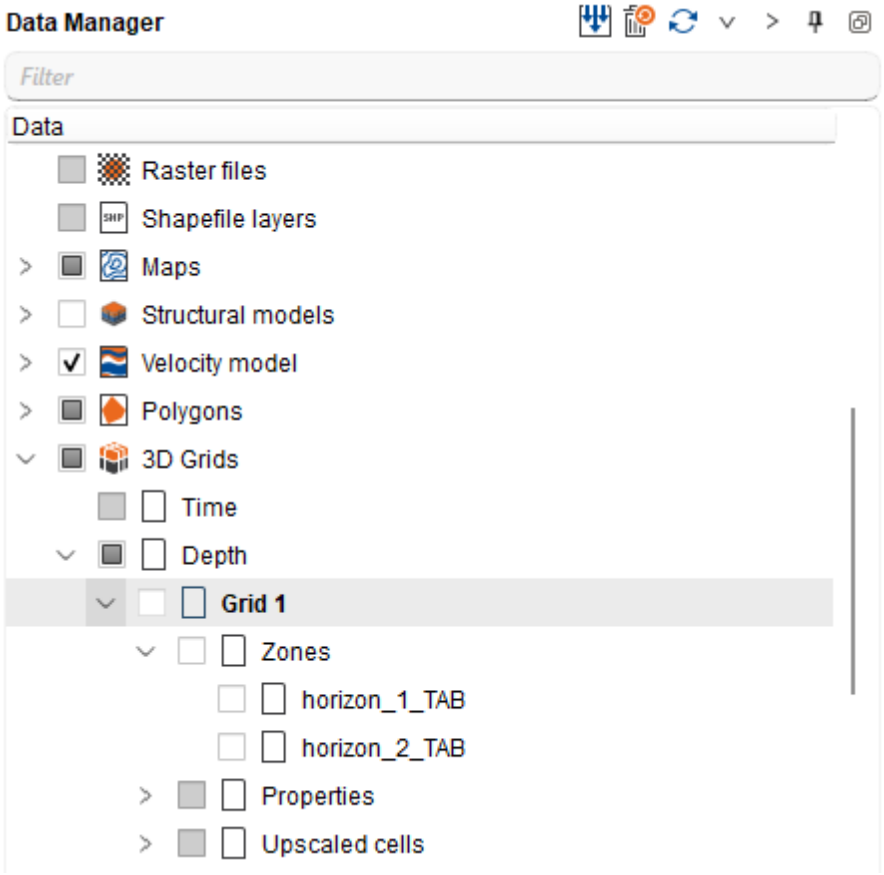
Afterward, choose the **Propagation type** and set the **Propagation distance**.

The **Property propagation parameters** window fills every grid cell the Upscale Log Data step left empty, zone by zone. Its **Source upscaled curve** comes from the grid's **Upscaled cells** folder, and the **Property name** defaults to that curve without the _U suffix (for example NPHI_U becomes NPHI).
The **Modeling type** drives the **Propagation type** list:

- **Petrophysical** — ABOS, Kriging or Stochastic.
- **Facies** — Voronoi or Stochastic.

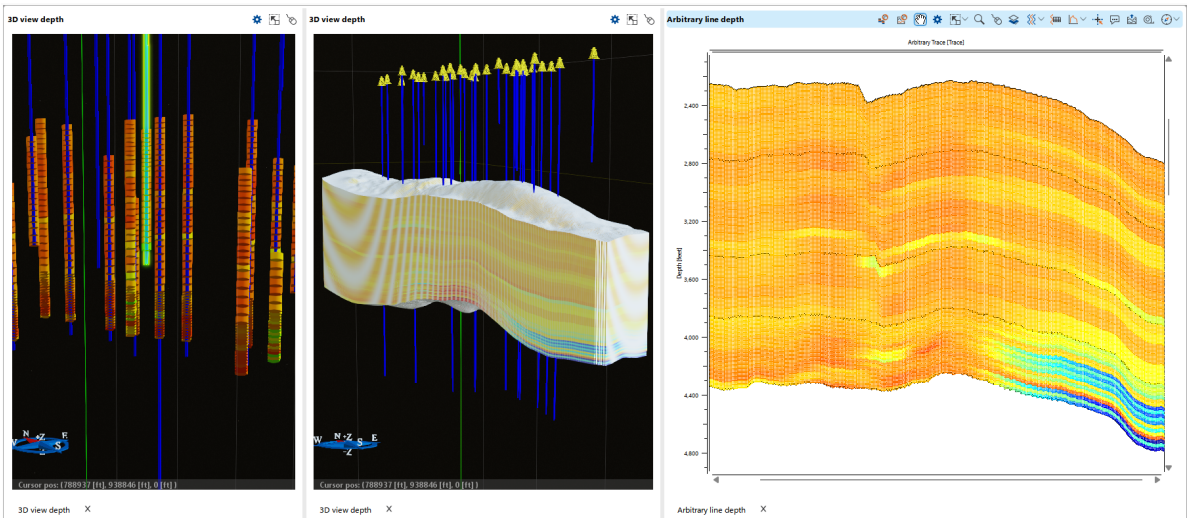
Propagation distance caps the neighbour-search radius. A **Trend (2D map)** can bias the result toward a chosen map, weighted by **Trend weighting, %** (0 % disables the trend).
Random seed can be left on auto or fixed to a value for reproducible stochastic runs.
When the propagation uses Kriging or a stochastic method, a variogram block becomes available (**Use variogram**, **Model** — Spherical, Exponential or Gaussian — **Sill**, **Nugget**, **Azimuth**, **Major distance** and **Minor distance**); facies stochastic runs additionally expose a connectivity filter.
Use **Edit per-zone settings...** to vary the algorithm, distance, trend and variogram per zone before running; each zone is then propagated using only its own upscaled cells, so no data crosses zone boundaries. **Save to workflow** stores the settings as a workflow step.
The propagated result is written to the grid's **Properties** folder and records its source curve, per-zone settings and random seed. Right-click a propagated property and choose **Show propagation settings** to reopen this wizard pre-filled with the recorded values.
Every grid carries a **Zones**, a **Properties** and an **Upscaled cells** folder under its main folder in the Data Manager, listed even while they are still empty. You can access a context menu for managing these objects by right-clicking on them.

© 2026 Geomage 2003 LTD



You can visualize the grid in **3D depth view** as a complete cellular model, display grid cells along wells, and also view the grid in the section view for detailed analysis.
Visualization settings are available in the Visual Settings 206

© 2026 Geomage 2003 LTD



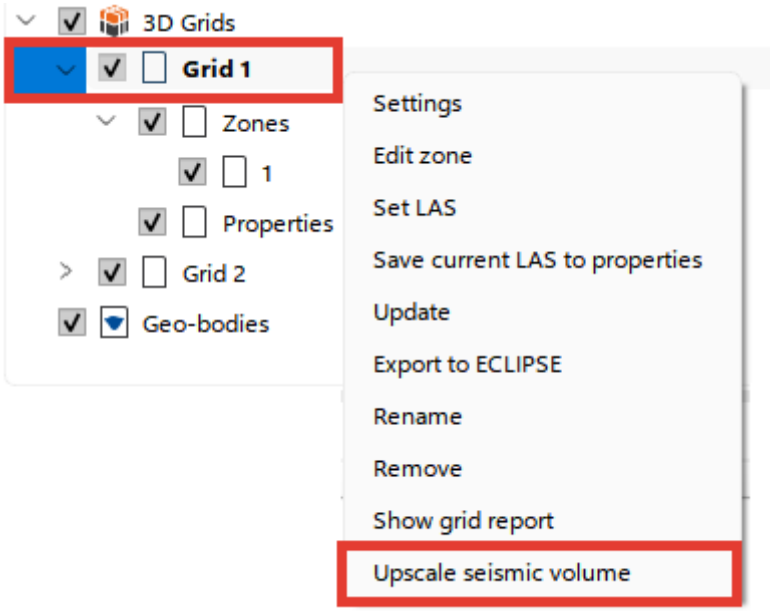
9.16.1 Seismic Data Upscaling

Seismic upscaling in g-Space allows users to integrate seismic data into a 3D grid, converting it into properties that can be used for geological and reservoir modeling.

Open the Context Menu

- Navigate to the **Data Manager** and locate the grid you wish to upscale seismic data for (e.g., "Grid 1").
- Right-click on the selected grid to open the context menu.
- Select the **"Upscale Seismic Volume"** option.

© 2026 Geomage 2003 LTD

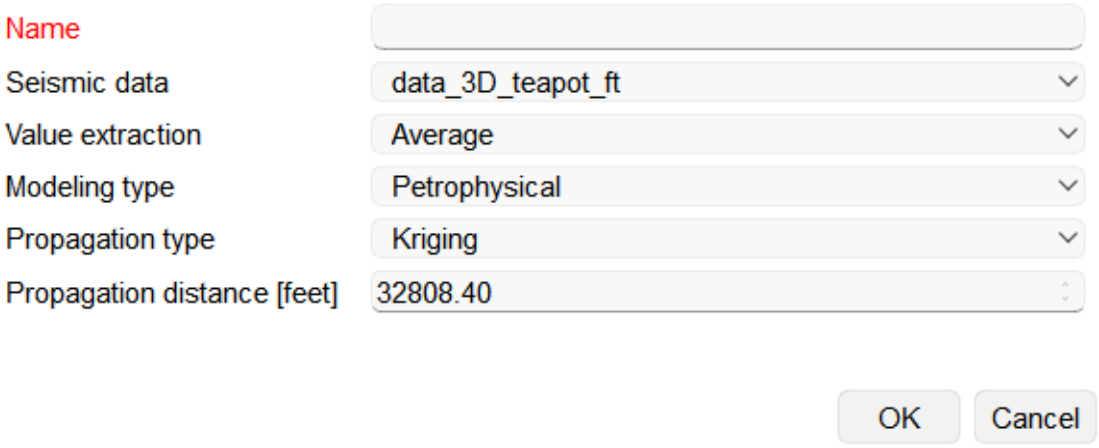


Configure Upscaling Parameters

A pop-up window titled **"Upscale Seismic Volume"** will appear. Fill in the required fields:

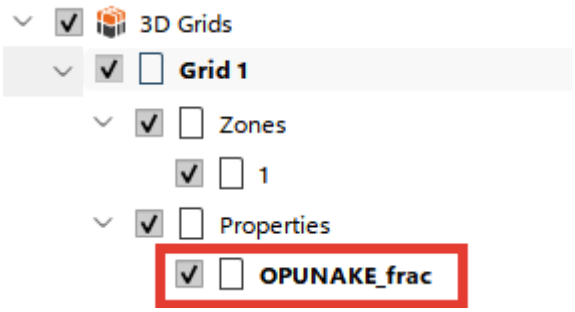
- Name: Enter a name for the seismic property to be created (e.g., "Fractures").
- Seismic Data: Choose the seismic volume from the dropdown list.
- Value Extraction: Select the method to calculate values from seismic data. Options include:
  - **Average:** Computes the mean value over the selected seismic cells.
  - **RMS**
  - **Minimum/Maximum:** Selects the lowest or highest value, respectively, from the seismic data.
  - **Median:** Computes the middle value, reducing the effect of outliers.
  - **Most common**
- Modeling Type: Specify the type of modeling as Petrophysical or Facies, depending on whether the property is scalar or categorical.
- Propagation Type: Choose the method used to distribute the seismic property across the grid. For Petrophysical modeling the options are ABOS and Kriging; for Facies modeling the options are Voronoi and Stochastic.
- Propagation Distance: Define the maximum distance (in feet or meters) for propagating the seismic property.

© 2026 Geomage 2003 LTD



Execute Upscaling

- After configuring the parameters, click "OK" to start the seismic upscaling process.
- The system will populate the 3D grid with the seismic property based on the specified settings.



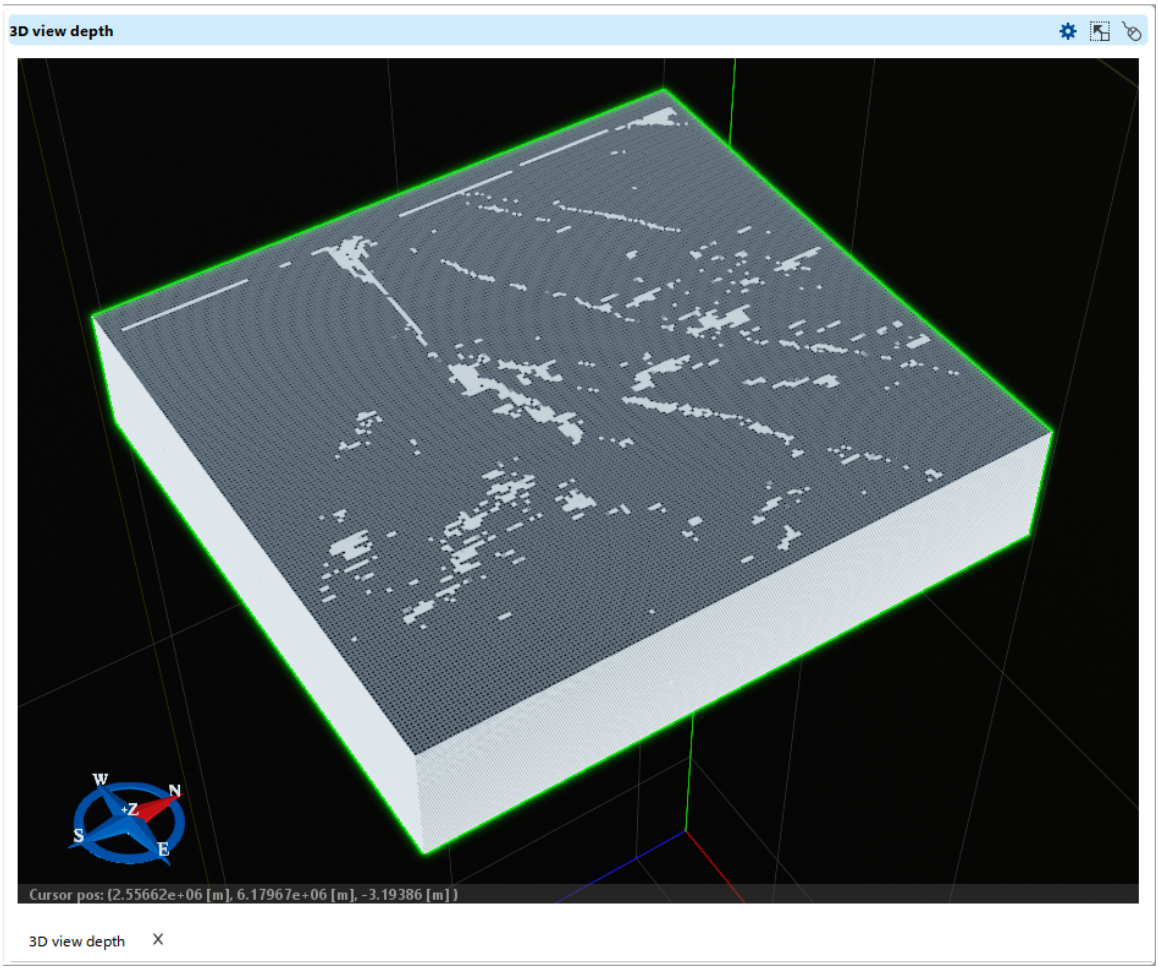
Review and Manage Properties

- Once the process is complete, the new seismic property will be added to the Properties folder under the grid in the **Data Manager**.

Visualize the Upscaled Property

- Open the 3D Depth View or Section View to analyze the grid with the newly upscaled seismic property.
- Adjust visualization settings in the Visual Settings panel to customize the display.

© 2026 Geomage 2003 LTD



9.16.2 Upscale 3D Grid

Upscale 3D Grid creates a new, coarser 3D grid from an existing structured grid by combining several fine cells into one larger cell. It is used to reduce the cell count of a 3D grid before further modeling or simulation while preserving the essential distribution of a property.
The operation always produces a **new** upscaled grid in the Data Manager — the original grid is never overwritten. Upscaling is available for structured grids (S-Grids) only; it is not supported for tetrahedral grids.

Purpose

The tool is designed for:

- Reducing the resolution (cell count) of a 3D grid to speed up modeling, visualization and simulation

© 2026 Geomage 2003 LTD

- Combining several fine cells into one coarser cell along the I, J and K grid directions
- Recalculating property values for the new, larger cells using a chosen statistical rule
- Preparing a lightweight grid as part of a larger grid-preparation workflow

Requirements

- A structured 3D grid (S-Grid) available in the Data Manager. Tetrahedral grids are not supported.
- At least one grid property to upscale (upscaling is launched from a property of the grid)
- Upscaling factors that produce a coarser grid — at least one of the I, J or K values must be greater than 1

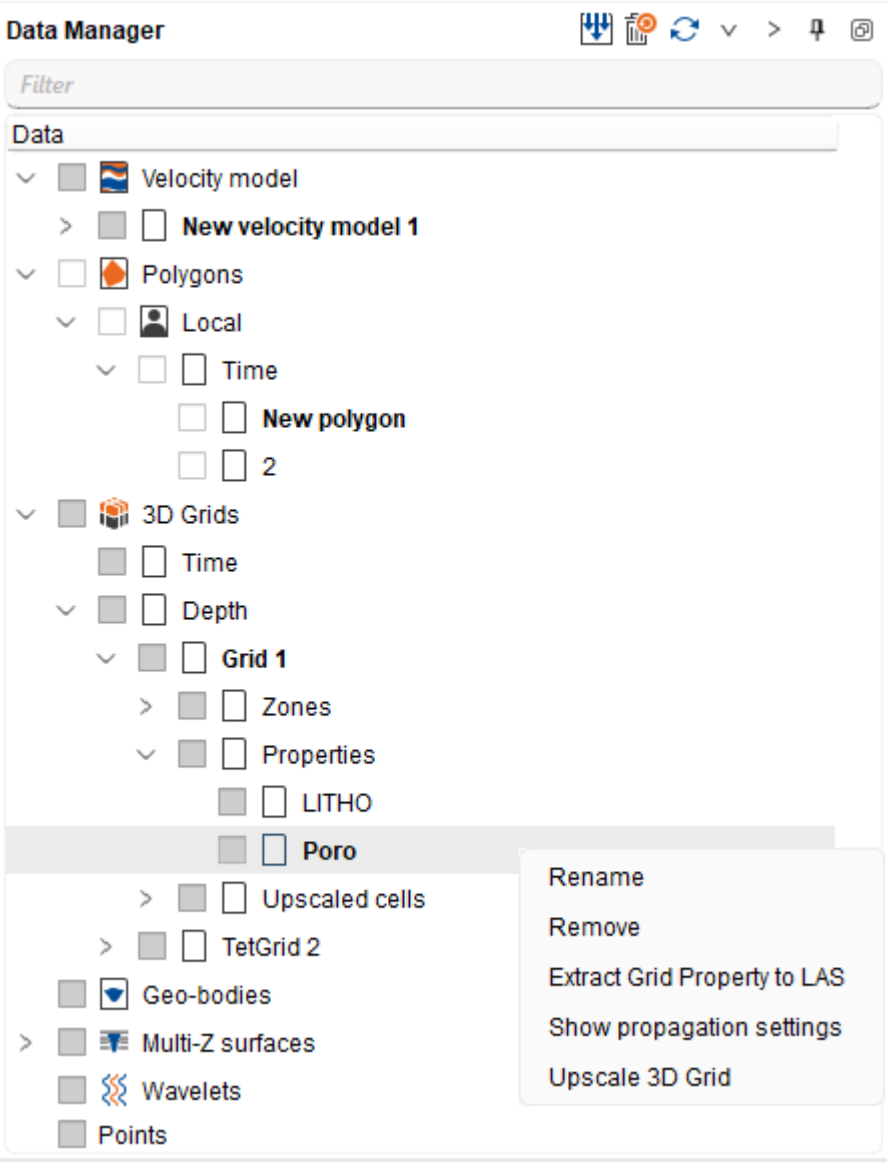
Launching the Wizard

The wizard is opened from the property context menu of a 3D grid in the Data Manager.

1. In the **Data Manager**, expand the target 3D grid and open its **Properties** folder.
2. Right-click the property you want to upscale.
3. Select **Upscale 3D Grid** from the context menu.

The right-clicked property is used as the input, and the resulting upscaled grid contains that single property. The wizard opens as a dialog titled **Upscale 3D Grid - <grid> / <property>**.

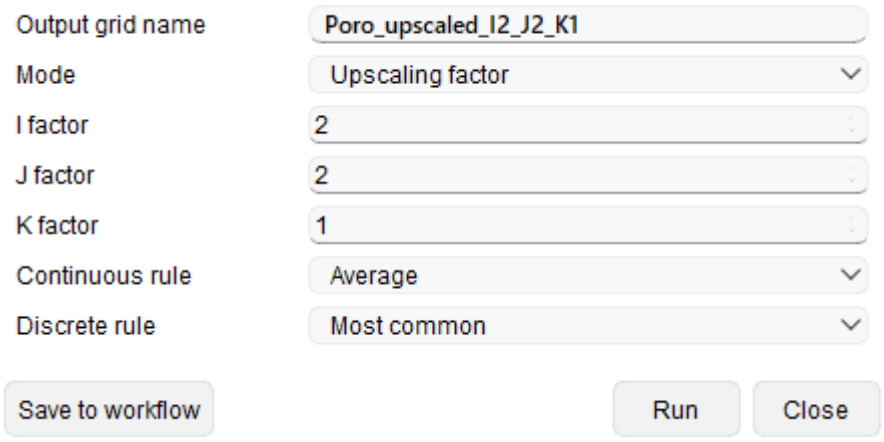
© 2026 Geomage 2003 LTD



Step-by-Step Guide

All settings are collected in a single dialog. Configure the output name, the target resolution and the property rules, then run the operation.

© 2026 Geomage 2003 LTD



Output Grid Name

Output grid name — the name of the new upscaled grid. A default name is proposed from the source property name with an `_upscaled_I<i>_J<j>_K<k>` suffix that reflects the chosen factors (for example `Porosity_upscaled_I2_J2_K1`). The name must not be empty and must not match an existing grid; a unique name is generated automatically if a clash occurs.

Output Resolution

Mode — defines how the new cell size is specified:

- **Upscaling factor** — how many original cells are combined into one larger cell along each direction.
- **Target cell size** — the desired output cell size; the factor is derived automatically from the input cell size.

When **Upscaling factor** is selected, set the per-direction factors:

- **I factor** — cells combined along the first horizontal grid direction (default 2).
- **J factor** — cells combined along the second horizontal grid direction (default 2).
- **K factor** — cells combined vertically, layer to layer (default 1).

When **Target cell size** is selected, set **I target cell size**, **J target cell size** and **K target cell size** instead. The visible controls switch automatically with the selected mode.

The **K factor** is kept at 1 by default because vertical layering is usually geologically meaningful. At least one of the I, J or K factors must be greater than 1, otherwise the grid would not be coarsened.

© 2026 Geomage 2003 LTD

Property Upscaling Rules

The value of each new, larger cell is calculated from the fine cells it covers. The rule that is applied depends on the property type — continuous (Petrophysical) or discrete (Facies):
Continuous rule — applied to continuous properties:
- **Average** — mean of the covered cell values (default).
- **Median** — middle value, reducing the effect of outliers.
- **Minimum** — lowest covered value.
- **Maximum** — highest covered value.

Discrete rule — applied to discrete (categorical) properties:
- **Most common** — the most frequent code among the covered cells (default).
- **Dominant** — the prevailing code among the covered cells.

Running the Upscaling

Press **Run** to create the upscaled grid immediately, or **Save to workflow** to store the operation as a workflow step (see below). Use **Close** to dismiss the dialog. When Run finishes, the new grid appears in the Data Manager 124.

Adding to a Workflow

The Upscale 3D Grid operation can be added to a Workflow 535 so it can be reused and re-run later:
1. Configure all settings in the wizard as usual.
2. Press **Save to workflow** instead of **Run**. A workflow step of type **Upscale 3D grid** is appended, with all selected parameters (mode, factors or target cell sizes, and property rules) stored inside it.
3. Re-run the step from the workflow at any time, or reopen it to edit the parameters.

This makes the operation a repeatable building block in a larger grid-preparation workflow.

Results

- A **new** upscaled 3D grid is added to the Data Manager 124; the original grid is unchanged.
- The grid geometry is coarsened according to the selected factor or target cell size, with the cell step scaled by the factor along each direction.
- Continuous properties are recalculated with the selected continuous rule; discrete properties are recalculated with the selected discrete rule.
- Undefined cells remain undefined when a new cell has no valid input values; otherwise the output value is computed from the valid covered cells.

© 2026 Geomage 2003 LTD

- Multi-zone grids are supported — each zone is upscaled independently, preserving the zone structure.
- The upscaled grid and its property can be displayed in the 3D View 244 and section views, and are saved and restored with the project.

Recommendations

- Keep **K factor** at 1 unless you specifically want to merge layers vertically, so the original vertical layering is preserved.
- Start with a modest lateral factor (for example I = 2, J = 2) and review the result before coarsening further.
- Use **Average** or **Median** for continuous petrophysical properties, and **Most common** for facies and other categorical properties.
- Use **Target cell size** when you need the output grid to match a specific cell size rather than a fixed combining factor.

See Also

- 3D Grids 169
- Creating a 3D grid 765
- Properties modeling 776
- Seismic Data Upscaling 782
- Workflows 535

9.17 Volumetric calculations

Volumetric calculations combine a reservoir boundary with rock and fluid properties to estimate effective reservoir volume, oil, gas or CO2 volumes. Start from the **Reservoir** tab.

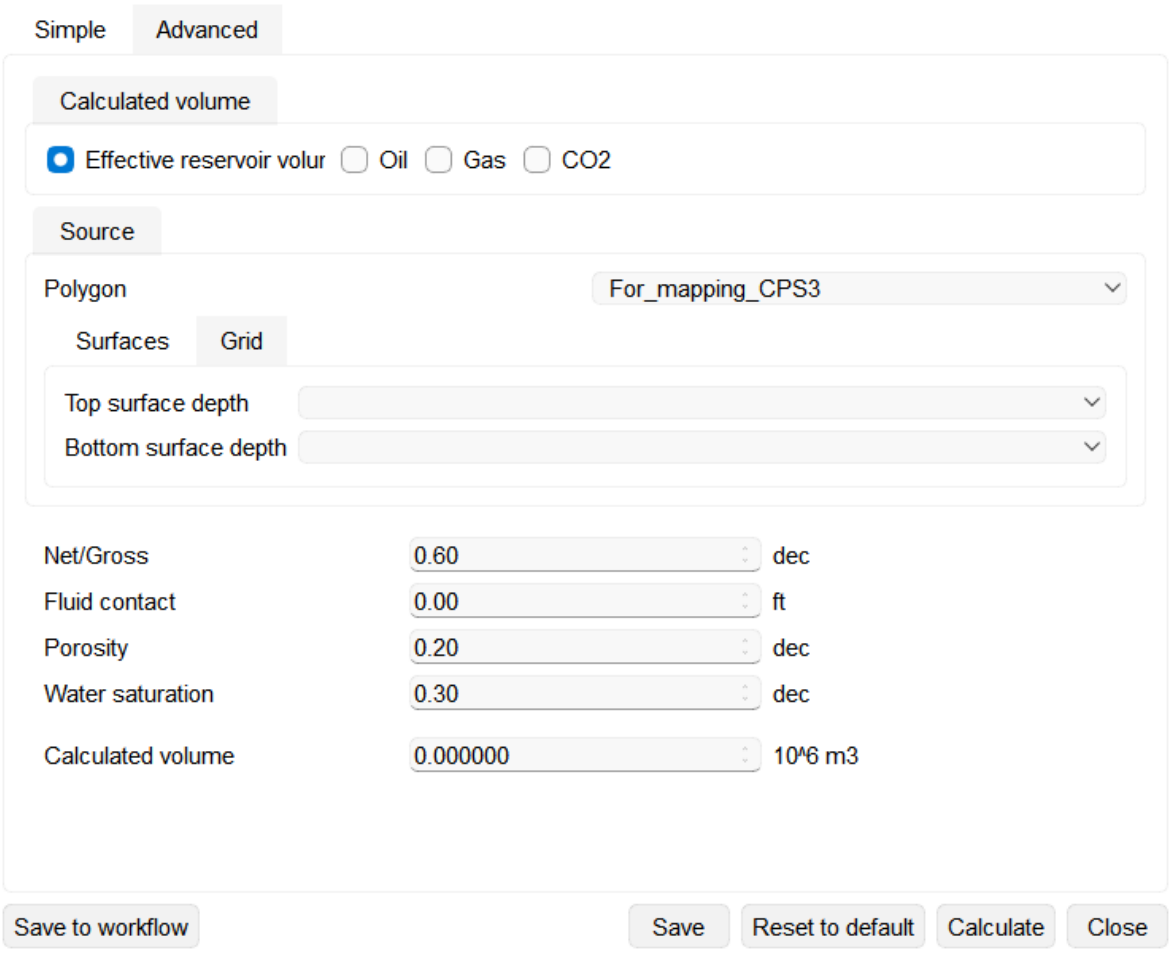
Calculate gross rock volume

In the **GRV** group, choose a **Polygon** and the first **Surface** as the top depth map. Choose the second **Surface** when a bottom depth map is needed, or leave it set to **Not required** when the fluid contact defines the lower boundary.
Enter the **Fluid contact** depth and click **Calculate gross rock volume**. The ribbon displays the resulting **GRV from Fluid contact** and **Fluid contact area**.
See Reservoir bar 122 for the current ribbon layout and command locations.

Calculate reserves

© 2026 Geomage 2003 LTD

In the **Reserves** group, click **Calculate reserves**. The wizard provides **Simple** and **Advanced** modes for **Effective reservoir volume**, **Oil**, **Gas** and **CO2**.



The image shows the **Simple Effective reservoir volume** layout before the top and bottom depth maps are selected.

Simple mode

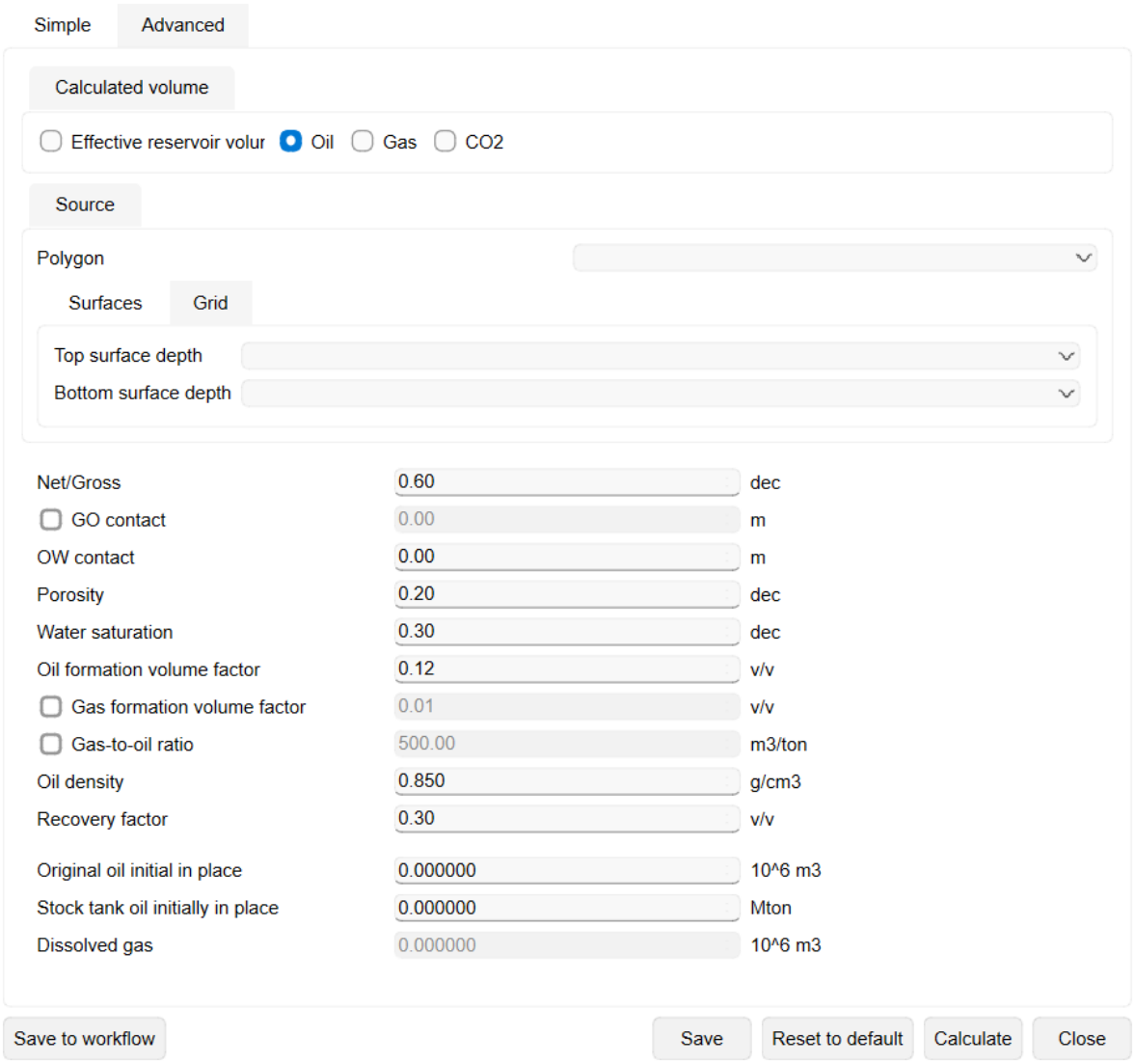
1. On the **Simple** tab, select the required **Calculated volume**.
2. Choose the bounding **Polygon**. On the **Surfaces** tab, select both **Top surface depth** and **Bottom surface depth**. Alternatively, open the **Grid** tab and select an existing project **Grid**.
3. Enter the rock, contact and fluid parameters displayed for the selected volume type.
4. Click **Calculate**. The result is displayed in the wizard, which remains open for further changes or calculations.

Oil parameters

Selecting **Oil** replaces the parameters below the **Source** area with the oil parameters. Each one opens with a standard value: **Net/Gross** 0.60 dec, **Porosity** 0.20 dec, **Water saturation** 0.30

© 2026 Geomage 2003 LTD

dec, **Oil formation volume factor** 0.12 v/v, **Oil density** 0.850 g/cm3 and **Recovery factor** 0.30 v/v.



OW contact sets the lower depth limit of the calculated volume. Select the optional **GO contact** checkbox, which is cleared when the wizard opens, to add an upper depth limit so that only the interval between the gas-oil and the oil-water contact is used. While **GO contact** is cleared, no upper limit is applied and the volume is bounded by the selected top surface or grid down to **OW contact**.

Two further checkboxes are also cleared when the wizard opens: **Gas formation volume factor** 0.01 v/v and **Gas-to-oil ratio** 500 m3/ton. Selecting a checkbox makes its value available, and the read-only **Dissolved gas** result stays unavailable until both are selected.

The read-only boxes under the parameter list report **Original oil initial in place** in 10^6 m3, **Stock tank oil initially in place** in Mton and **Dissolved gas** in 10^6 m3. On this page, **Reset to default** restores the standard values and clears the three optional checkboxes, but keeps the **GO contact** and **OW contact** depths you have entered.

© 2026 Geomage 2003 LTD

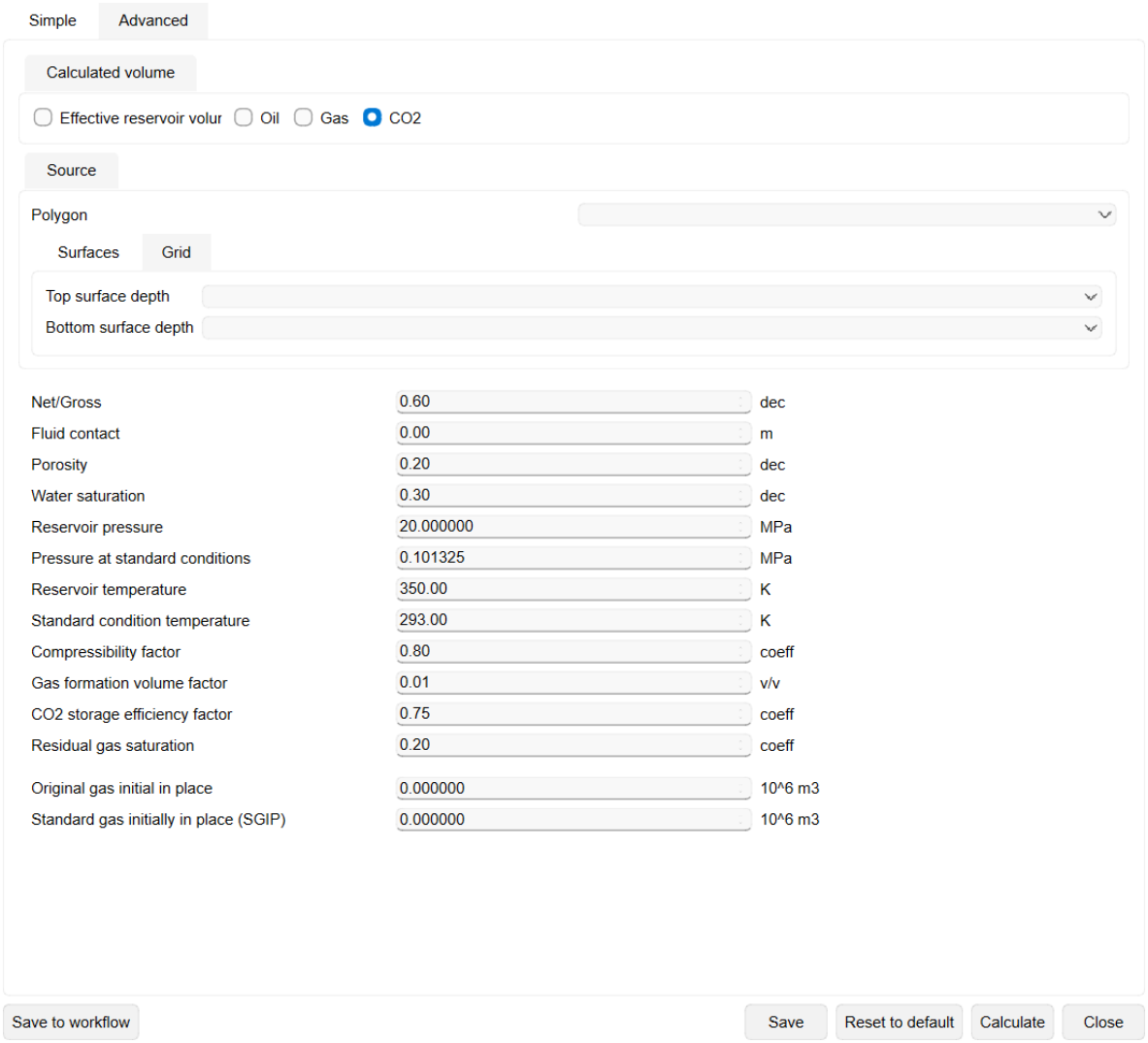
Gas parameters

Selecting **Gas** replaces the parameters below the **Source** area with the gas parameters. Each one opens with a standard value: **Net/Gross** 0.60 dec, **Fluid contact** 0.00 m, **Porosity** 0.20 dec, **Water saturation** 0.30 dec, **Reservoir pressure** 20 MPa, **Pressure at standard conditions** 0.101325 MPa, **Reservoir temperature** 350 K, **Standard condition temperature** 293 K, **Compressibility factor** 0.80 coeff and **Gas formation volume factor** 0.01 v/v.
The two read-only boxes under the parameter list report **Original gas initial in place** and **Standard gas initially in place (SGIP)**, both in 10^6 m3.

CO2 parameters

Selecting **CO2** replaces the parameters below the **Source** area with the CO2 storage parameters. Each one opens with a standard value: **Net/Gross** 0.60 dec, **Fluid contact** 0.00 m, **Porosity** 0.20 dec, **Water saturation** 0.30 dec, **Reservoir pressure** 20 MPa, **Pressure at standard conditions** 0.101325 MPa, **Reservoir temperature** 350 K, **Standard condition temperature** 293 K, **Compressibility factor** 0.80 coeff, **Gas formation volume factor** 0.01 v/v, **CO2 storage efficiency factor** 0.75 coeff and **Residual gas saturation** 0.20 coeff.
Change the values that differ for the storage site before you calculate.

© 2026 Geomage 2003 LTD



The **CO2** page reports the same two read-only results as **Gas**: **Original gas initial in place** and **Standard gas initially in place (SGIP)**, both in 10^6 m3. They stay at zero until **Calculate** runs, and **Reset to default** restores the standard values on the page while keeping the **Fluid contact** depth you have entered.

Advanced mode

The **Advanced** tab performs a stochastic estimate and reports percentile results. It uses the polygon selected on the **Simple** tab together with the depth-map limits configured in **Surface maps**.

- Set **Number of Realizations** and **Smoothing radius (grid nodes)**. Enable **Fixed seed** when the same random sequence must be reused.
- For **Top Surface Depth**, choose **Upper limit** and **Lower limit** maps. Do the same for **Bottom Surface Depth**, or clear **Active** when no bottom surface is used.

© 2026 Geomage 2003 LTD

- Define the distributions for the parameters shown for the selected volume type. Enable **Correlation matrix** when correlations between inputs must be included.
- The **Calculated volume (percentiles)** panel reports P99, P90, P50, P10 and P1. **Sensitivity analysis** is available for Effective reservoir volume only.

Run, save or add to a workflow

- **Save to workflow** adds a **Calculate reserves** job with all current settings to the current workflow 535. It does not run the calculation, and the wizard remains open.
- **Save** stores the current wizard settings and closes the wizard.
- **Reset to default** restores the standard parameter values for the active calculation mode.
- **Calculate** runs the calculation without closing the wizard.
- **Close** closes the wizard without saving the current changes.

When a saved **Calculate reserves** job is opened from Workflows 535, its saved mode, source and parameters are restored. The footer then shows only **Save** and **Cancel**: use **Save** to update the job or **Cancel** to leave it unchanged. Executing the workflow runs the calculation with the stored settings.

Create a geo-body

To create geometry from the current GRV inputs, use **Create geo-body from gross rock volume** in the **Geo-body** group. For the broader structural-model workflow, use **Create 3D geological model** and see Geological modeling 758.

See Also

- Reservoir bar 122
- Workflows 535
- Geological modeling 758

9.18 Multi-user mode

Multi-User mode in g-Space enables collaborative project management by allowing users to work on shared projects using a centralized database.
This mode ensures that multiple team members can connect to the same project database, manage seismic data, and synchronize updates in real-time.

The main functions involved are setting up the database connection, receiving and sending data, and managing synchronization settings.

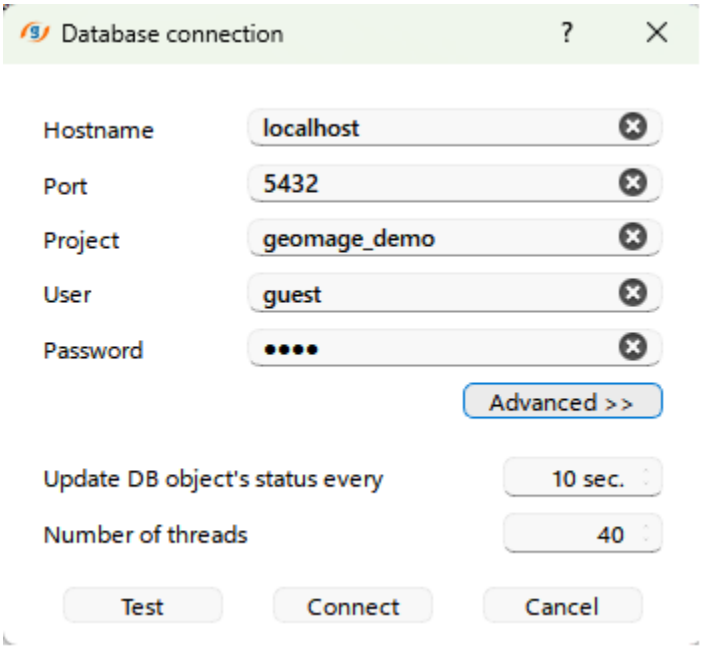
© 2026 Geomage 2003 LTD

1. Set Connection

To enable multi-user functionality, you first need to establish a connection with the remote database. This can be done via the **Set Connection** window. The fields that need to be filled include:

- **Hostname**: The server address (e.g., localhost or a remote server IP)
- **Port**: Database connection port (e.g., 5432)
- **Database**: The name of the project database to connect to.
- **User**: The username to access the database.
- **Password**: The user's password for authentication.

Additionally, advanced options such as **Number of Threads** and the interval for **Updating DB object status** can be configured here. Once the details are set, users can **Test**, **Connect**, or **Cancel** the connection.



2. Receive from DB

Once connected to the remote project, the **Receive from DB** function allows users to pull data from the shared database into their local environment. This ensures that you are working with the latest version of the project's seismic, well, and horizon data, among others.

Data can be received for specific objects such as **Volumes**, **Lines**, **Horizons**, **Faults**, **Wells**, **Checkshots**, and **LAS** files. It ensures that the local project is synchronized with the latest data on the database.

© 2026 Geomage 2003 LTD

3. Send to DB

The **Send to DB** function allows users to push local changes to the database. This includes any modifications or new data added to the local project, such as edited horizons, seismic volumes, or well logs.

This feature is essential for maintaining data integrity across users, ensuring that the latest modifications are available for other team members working on the same project.

4. Sync Settings

The **Sync Settings** feature provides more granular control over data synchronization. Users can define how often their local project synchronizes with the database and specify which data types or project elements should be included in each synchronization cycle.

5. Delete DB

The **Delete DB** feature allows users with the necessary permissions to remove the project database entirely. This should be used cautiously, as it will permanently delete all data stored in the shared project database.

6. Data Manager Integration

Within the **Data Manager**, users can easily see which data files are stored locally and which are part of the shared remote database:

- **Local Folders**: These are the files saved on your local project and are not synchronized with the database.
- **Database Folders**: Represent the shared project data stored in the database. Users can perform actions like **Receive** or **Commit** by right-clicking for specific data objects such as **Volumes**, **Lines**, **Horizons**, **Faults**, **Wells**, **Checkshots**, and **LAS**.

These options ensure flexibility and control over how data is managed and shared across multiple users in the project.

© 2026 Geomage 2003 LTD

9.19 Structural modeling

Structural modeling is the process of building a three-dimensional geological framework from interpreted horizons and faults. In g-Space, the Structural Modeling wizard takes interpreted horizons – or the geological objects organized in the Conceptual Model 191 – assigns conformity relationships and fault constraints, and produces a consistent set of TIN surfaces that honor all input data. The resulting structural model serves as the foundation for property modeling, volumetric calculations, and reservoir characterization.

Purpose

The Structural Modeling wizard is designed for:

- Building depth- or time-domain TIN surfaces from interpreted horizons or Conceptual Model objects
- Combining available Points, Maps, Multi-Z surfaces and Markers as Soft or Hard inputs
- Applying Depositional, Unconformity or Intrusion roles in the selected stratigraphic order
- Optionally honoring a fault model and limiting the model with a border polygon
- Generating horizon surfaces, layers and fault blocks for section and 3D views

Requirements

- At least one horizon or Conceptual Model object with an available Points, Maps, Multi-Z or Markers input
- Optionally, a Conceptual Model 191 that organizes the input objects and their conformity roles
- Optionally, a fault model for fault-constrained modeling
- Optionally, a border polygon to limit the modeling area

Launching the Wizard

Launch methods:

- On the Modelling 100 tab, in the **Structural Modeling** group, click **Structural modeling**
- In Data Manager 124, right-click the Structural models 163 branch, its user folder, or a **Depth/Time** folder and choose **Create new**. Starting from a domain folder presets that domain

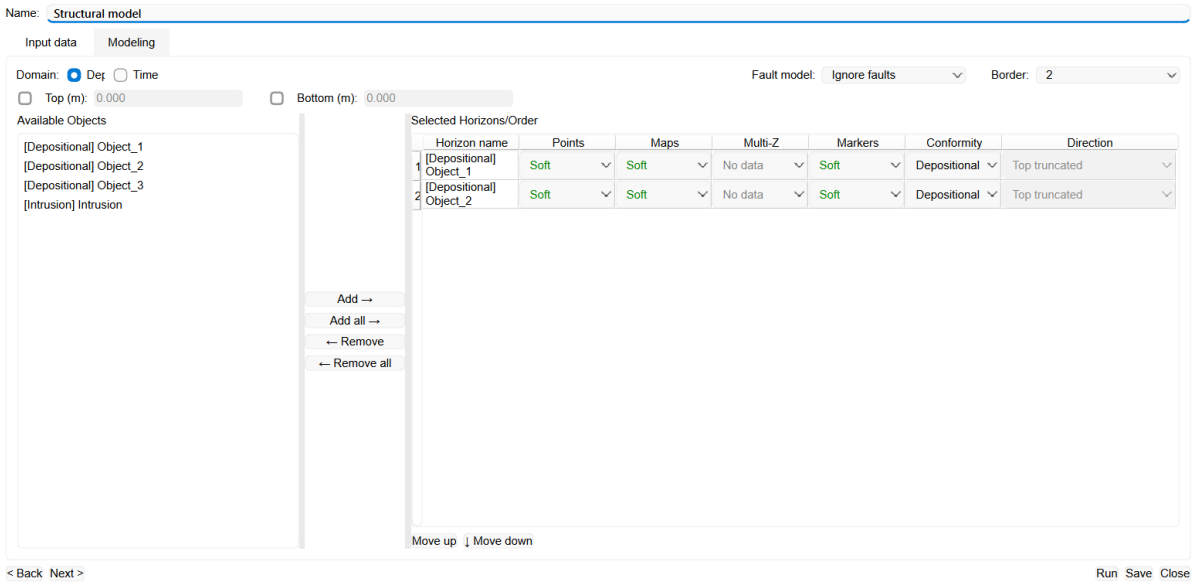
© 2026 Geomage 2003 LTD

The wizard has a **Name** field and two tabs: **Input data** and **Modeling**. A new model receives a unique default name; the name must remain non-empty and unique. Use **Back** and **Next** to move between the tabs.

Step-by-Step Guide

Input Data Tab

This tab is where you select horizons, define their stratigraphic order, assign conformity relationships, and specify fault constraints.



1. Select the domain

- Choose between **Depth** (default) and **Time** domain using the radio buttons at the top of the dialog
- The domain determines which horizon data (depth maps or time maps) will be used for modeling

2. Select fault model and border

- **Fault model:** The dropdown lists **Ignore faults** followed by the available fault models. When the Conceptual Model 191 has a fault model linked to it, that model is preselected here; you can still pick another one, or choose **Ignore faults** to model unfaulted surfaces
- **Border:** The dropdown lists **No border** followed by the available border polygons. When the project contains polygons, the first one is selected by default; choose **No border** to model

© 2026 Geomage 2003 LTD

the full extent of the input data. If the project has no polygon at all, a warning is shown when you run the model and a border is created automatically (see **Running the Model** below)

When a horizon takes its shape directly from a **Hard** Multi-Z surface (see step 4), give every reverse fault in the selected fault model the **Fault Type Reverse** on the **Fault Inputs** tab of the Fault Modeling wizard (see Fault model 181): only that value keeps each piece of the split surface on the side of the fault it came from, so a folded surface that crosses the fault does not keep parts on the wrong side. Every other fault type splits such a surface by fault topology alone, and any piece whose side cannot be resolved is left out of the model without a message.

When a fault model is used, each modeled horizon is smoothed where it meets the fault surfaces, so that it approaches a fault evenly instead of reproducing the small vertical steps of the fault surface. The smoothed strip along the fault is as wide as the **Fault clear radius** value in the **Map interpolation** group of Module properties 197, and inside it the height of the horizon is continued from the surrounding part of the same piece of the surface, so close to a fault the surface follows that trend rather than its own input data. Only heights are adjusted, so the surface keeps the outline the fault split gives it; a **Fault clear radius** of 0 leaves the horizon exactly as the split produced it.

3. Select horizons or objects

The left panel provides the input list. When the Conceptual Model 191 contains objects for the selected domain, the panel is titled **Available Objects** and lists those objects, each prefixed with its conformity (**[Depositional]**, **[Unconformity]** or **[Intrusion]**); only objects whose data matches the selected domain are shown. When there are no conceptual objects for the domain, the panel falls back to **Available Horizons** and lists the project horizons directly.
Use the **Add** → and ← **Remove** buttons to move items between the available list and the selected table. Use **Add all** → to include everything at once, or ← **Remove all** to clear the selected table. Multiselection works the same way as in the Fault Modeling wizard. When the panel lists **Available Objects**, every object is tracked as its own row: **Add** → and **Add all** → add each selected object as a separate, independently configurable row, two objects that reference the same horizon remain two separate rows, and the same object cannot be added twice.

4. Configure the selected horizons table

The right panel shows the selected horizons with columns for:

- **Horizon name** — the entry you added: in **Available Objects** mode the row shows the object label with its conformity prefix (**[Depositional]**, **[Unconformity]** or **[Intrusion]**); in **Available Horizons** mode it shows the horizon name
- **Points** — usage of horizon picks (No data / Soft / Hard)
- **Maps** — usage of horizon maps (No data / Soft / Hard)
- **Multi-Z** — choose **No data**, or select a named surface as a **Soft** or **Hard** input. Conceptual Model rows offer their assigned surface; in **Available Horizons** mode the list is not filtered by domain and offers every multi-Z surface in the project, so check yourself that the surface

© 2026 Geomage 2003 LTD

you pick belongs to the selected domain. When a row's only input is a **Hard** Multi-Z surface (**Points**, **Maps** and **Markers** are all **No data**), the picked surface mesh is used exactly as it is instead of being re-interpolated onto the modeling grid
- **Markers** — usage of well markers (No data / Soft / Hard)
- **Conformity** — stratigraphic role of the horizon/object: **Depositional**, **Unconformity** or **Intrusion**
- **Direction** — for unconformities, **Top truncated** or **Bottom truncated**. This cell is active only when Conformity is set to **Unconformity**

Use **Move up** and **Move down** buttons to set the correct stratigraphic order. Horizons should be ordered from shallowest (top) to deepest (bottom).

Data usage modes:
- **No data** — this data type is not used for the horizon
- **Soft** — data is used as a trend guide but the surface may deviate from it
- **Hard** — the surface is forced to pass exactly through these data points

Each usage cell is colour-coded for quick review: **No data** is grey, **Soft** is green and **Hard** is blue. Data types that are not available for a horizon or object appear as a disabled **No data** cell. For **Multi-Z** in **Available Horizons** mode that state is decided per domain rather than per horizon: the cell is enabled on every row as soon as the project contains at least one multi-Z surface of the selected domain, so a disabled cell there means the project has no multi-Z surface in that domain, not that this horizon has none.

A row whose **Conformity** is **Intrusion** is built as an intrusion body only when its **Multi-Z** cell is the row's only input and is set to **Hard** – the same direct Multi-Z combination described above, with **Points**, **Maps** and **Markers** all set to **No data** – and the selected surface is a closed shell that does not intersect itself. Any other **Intrusion** row is left out of the model completely: it produces neither an intrusion object nor a **TIN Maps** surface, and the skipped row is named by the warning reported after the run.

5. Set the vertical modeling limits

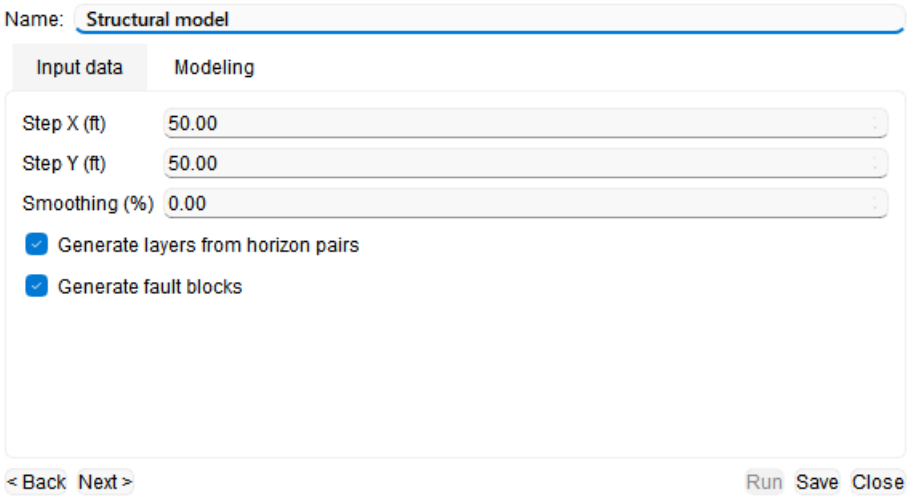
By default both limits are off and the model spans the full vertical extent of the selected data. Select the **Top** and **Bottom** check boxes to enable their value fields, whose labels use the current depth or time unit. When you enable a limit, its field is pre-filled from the selected horizons – **Top** from the shallowest and **Bottom** from the deepest data (the project datum is used for **Top** when nothing is selected) – and you can edit the value. An enabled limit also extends the model: a flat surface is built at that value across the modeled area, and the model gains an extra interval between that plane and the first selected horizon for **Top**, or below the last selected horizon for **Bottom**, so the model is no longer built only between the selected horizons. These flat surfaces are build artifacts rather than inputs: they are rebuilt from the current limit values on every **Run**, never accumulate, and are never listed as rows in the **Selected Horizons/Order** table when the model is reopened, so they cannot be configured or reordered. Clearing a check box disables its field and applies no limit on that side, so the model

© 2026 Geomage 2003 LTD

keeps its full extent there. Generated layer volumes are trimmed to the enabled plane as well as the modeled horizon and fault surfaces, while the flat **Top** and **Bottom** surfaces themselves lie on the plane and are neither clipped nor truncated by the **Conformity** and **Direction** rules; an intrusion body is left whole, so an enabled limit does not cut its shell.

Modeling Tab

This tab contains parameters that control the output grid resolution and surface smoothness.



Parameters:

- **Step X:** Grid spacing in the X direction, shown in the project horizontal unit (default: 50). A smaller value creates a finer grid and increases processing time
- **Step Y:** Grid spacing in the Y direction, shown in the project horizontal unit (default: 50)
- **Smoothing (%):** Surface smoothing from 0 to 100 (default: 0)
- **Generate layers from horizon pairs:** Enabled by default. Creates volume layers between consecutive modeled surfaces and stores them under **Layers**
- **Generate fault blocks:** Enabled by default. When a fault model is selected, creates separate blocks and stores them under **Fault Blocks**

Running the Model

After configuring the model, click **Run** to build a new model or update the open model. **Run** is available only when the **Name** is unique and non-empty and at least one selected horizon/object has a valid non-**No data** assignment. The wizard remains open after the run so that you can review or adjust its settings.

While the model is being built, the run is reported in the Progress View 221 as a task named **Build structural model** followed by the model name, with nested steps for interpolating the

© 2026 Geomage 2003 LTD

horizons, building the TIN model and publishing the results. You can stop the build there while any of those steps is still running; the stop is all-or-nothing, so no result objects are produced. A model that was being created for the first time is removed again and never appears under the Structural models 163 branch, while a model that was being rebuilt keeps its saved definition and must be run again to produce results.

If the project contains no polygon at all, a warning is shown before the model is built: "*No polygon border is available in the project. The modeling border will be created automatically from the largest area extent of the selected objects.*" The border is then generated automatically from the largest area extent of the selected horizons/objects.

Click **Save** to store the current model definition without running it. The saved definition includes its domain, ordered inputs, usage modes, conformity settings, limits, fault and border choices, and modeling parameters. Saving a model that has already been built keeps its results: its **TIN Maps**, **Faults**, **Intrusions**, **Layers** and **Fault Blocks** entries stay in the Data Manager and stay listed under **Structural model 3D** in the **Visual Settings** panel, and both are refreshed together so that they show the same set. **Run** remains the only rebuild, so the objects on display are still the ones produced by the last run, and the newly saved settings take effect only when you run the model again. Two changes carried by the same **Save** affect those results: switching the **Depth**/**Time** domain discards the ones built in the previous domain, so the model keeps its saved definition but shows no result objects until it is run again in the new domain, while a new **Name** renames the kept objects together with the model, so they stay grouped under it in both places. The wizard remains open; use **Close** when you are finished.

Results:

- The named model is stored in the Structural models 163 branch under its user and domain
- Built horizon surfaces appear under **TIN Maps** and can be displayed in section and 3D views
- When a fault model is used, its modeled surfaces appear under **Faults**
- Each accepted **Intrusion** row produces exactly one object under **Intrusions**, named after the model and the selected surface, and never a **TIN Maps** surface
- **Layers** and **Fault Blocks** are created when their corresponding Modeling options apply
- An enabled vertical limit adds its flat surface under **TIN Maps** as **Top** or **Bottom**, and, when **Generate layers from horizon pairs** is enabled, the extra interval appears under **Layers** named after its pair, as Top/<first horizon> or <last horizon>/Bottom

An intrusion shell is used exactly as you picked it: it is not re-interpolated onto the modeling grid, not smoothed by **Smoothing (%)**, not split by faults and not truncated by the **Conformity** and **Direction** rules. Every modeled horizon surface and every fault surface that meets the intrusion is cut back to its boundary, so those objects show a hole where the intrusion sits, and every generated layer volume the intrusion crosses has the intrusion volume subtracted, leaving a cavity inside the layer instead of geology continuing through the body. An **Intrusion** row is not a layer boundary. Layers are still generated between consecutive non-intrusion rows in the order, so an intrusion placed between two horizons does not split the enclosing layer in two: that layer keeps its own top and bottom pair and simply gains the

© 2026 Geomage 2003 LTD

cavity. **Generate layers from horizon pairs** therefore needs at least two non-intrusion rows to produce anything.
If a selected object cannot be built from its assigned inputs, the model is still created and a warning identifies the affected object after the run. The same warning reports a skipped intrusion row, and any horizon, fault or layer that had to be left out because an intrusion cut or volume subtraction could not be completed reliably.

Recommendations

- Start with a coarse grid (larger Step X/Y) for quick preview, then refine
- Use **Hard** constraint for well markers to ensure the model honors well data exactly
- Use **Soft** constraint for maps to allow the model to adjust where map data may be less reliable
- Set **Conformity** and, for unconformities, **Direction** carefully because they control how adjacent surfaces are truncated. This truncation is skipped for a pair in which either horizon takes its shape directly from a **Hard** Multi-Z surface (see step 4); grid-based horizons are truncated as usual, and rows set to **Intrusion** follow the intrusion path described under **Results**
- Define a border polygon to reduce computation time and focus on the area of interest
- Place an enabled vertical limit outside the horizon stack – **Top** above the shallowest selected data and **Bottom** below the deepest. A limit that lands inside the stack still builds, but the extra layer it adds is inverted where the plane crosses a horizon
- Review results in 3D view to verify fault-horizon intersections

See Also

- Conceptual Model 191
- Structural Models 163
- Horizon picking 614
- Fault picking 627
- Geological modeling 758
- Volumetric calculations 790
- Properties modeling 776
- Data Manager 124

9.19.1 Structural Uncertainty Analysis

Structural Uncertainty Analysis is a wizard that quantifies the positional uncertainty of a structural model 798 by generating an ensemble of perturbed realizations. Each realization rigidly displaces the model's horizons and faults by a random amount drawn from a per-element probability distribution, so the spread of the resulting surfaces represents the range of

© 2026 Geomage 2003 LTD

geologically plausible structural positions. The wizard runs in the depth domain only and is launched from the context menu of a Structural Model object in the Data Manager 124.

Purpose

The tool is designed for:

- Estimating the positional uncertainty of interpreted horizons and faults
- Producing an ensemble of equiprobable structural realizations from a single base model
- Building an uncertainty envelope for risk-aware volumetric and reservoir analysis
- Testing how much each element is allowed to move independently of the others

Requirements

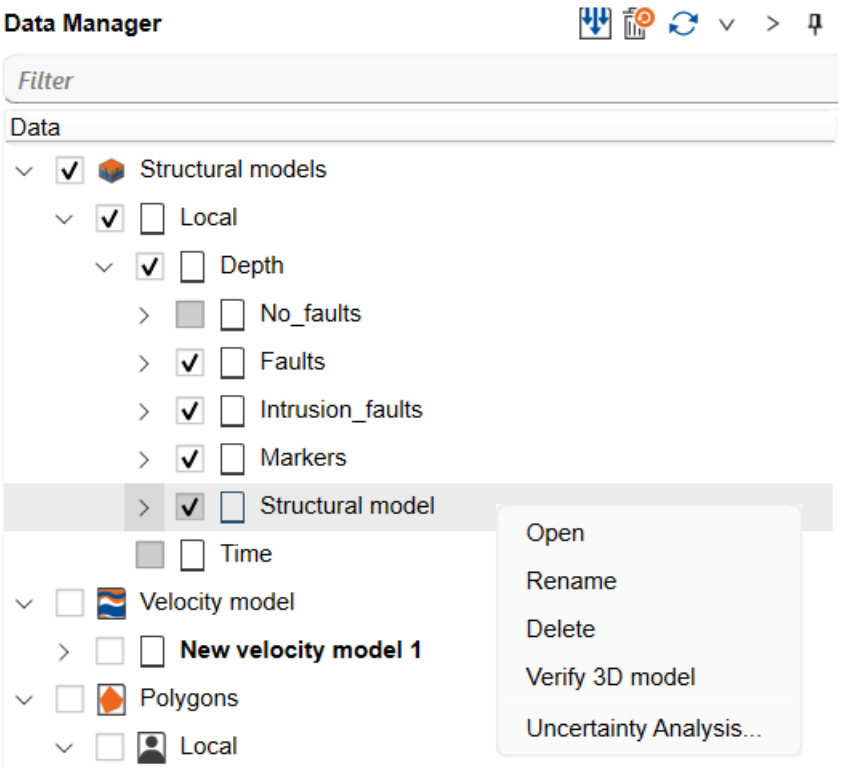
- A completed structural model 163 built in the **depth** domain (the analysis is not available for time-domain models)
- The base model must contain at least one horizon; faults are optional
- Well markers are optional — they are used only when the **Well markers** option is set to **Use** for an element

Launching the Wizard

Right-click a Structural Model object in the Data Manager 124 tree and choose **Uncertainty Analysis...** from the context menu.

- The command appears only for **depth-domain** models. It is not available for time-domain models or for individual realization leaves inside an existing ensemble.

© 2026 Geomage 2003 LTD



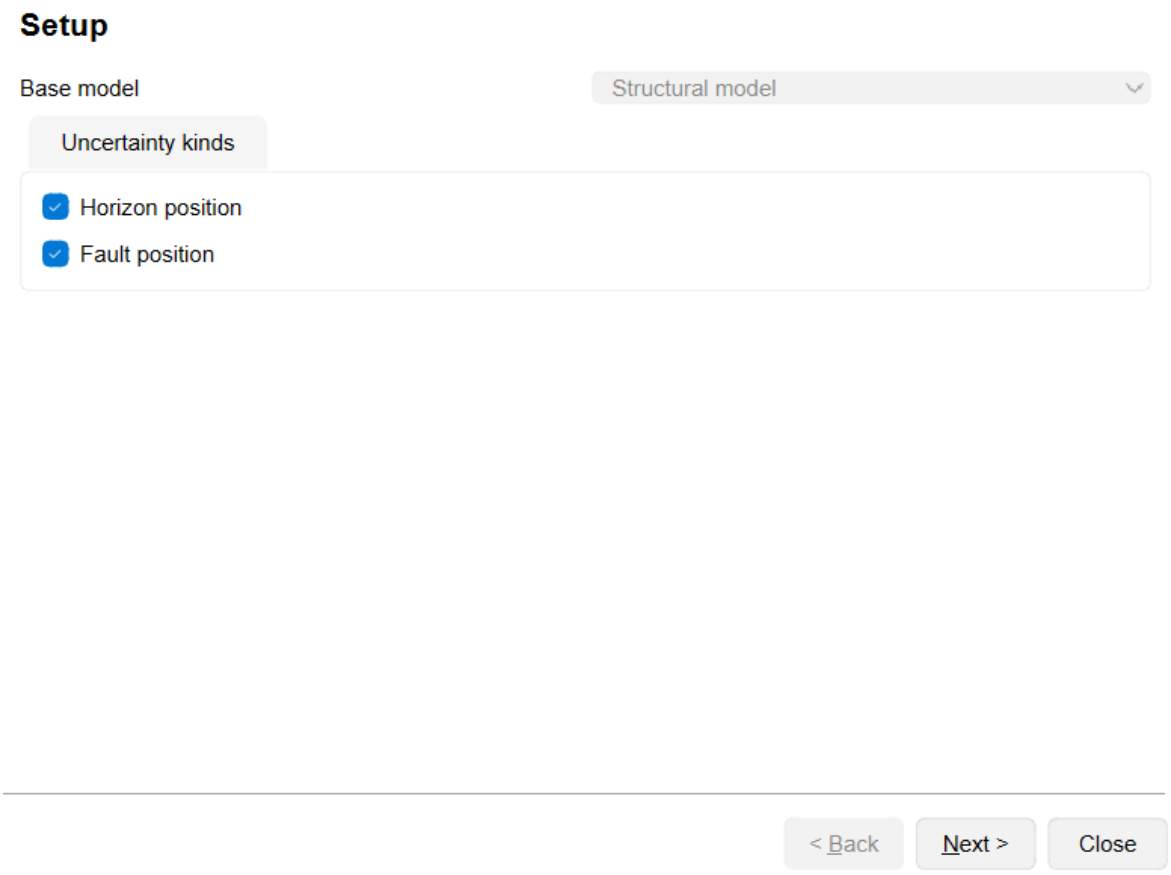
The wizard opens as a dialog window titled **Structural Uncertainty Analysis** with four pages (**Setup**, **Faults**, **Horizons**, **Run**). Navigate between pages using the **Back** and **Next** buttons. The **Faults** page is skipped automatically when **Fault position** is unchecked on the **Setup** page.

Step-by-Step Guide

Setup Tab: Select Model and Elements

The first page selects the base model and chooses which classes of element receive uncertainty.

© 2026 Geomage 2003 LTD



1. Confirm the base model

- The **Base model** dropdown is pre-filled with the model you right-clicked and is read-only — the analysis always runs on that model.

2. Choose the uncertainty kinds

In the **Uncertainty kinds** group, tick the element classes to perturb:

- **Horizon position** — perturbs horizon surfaces. Checked by default.
- **Fault position** — perturbs fault surfaces. Checked by default. When unchecked, the **Faults** page is skipped and no fault is perturbed.

Faults Tab: Fault Uncertainty

This page is shown only when **Fault position** is checked on the **Setup** page. It lists every fault in the model, one per row, and controls how each fault is allowed to move.

Global option

- **Horizontal displacement only** — when checked, fault movement is constrained to the horizontal (X–Y) plane; the fault-normal displacement is projected onto the map plane. Unchecked by default.

Fault table columns

- **Fault** — read-only name of the fault.

© 2026 Geomage 2003 LTD

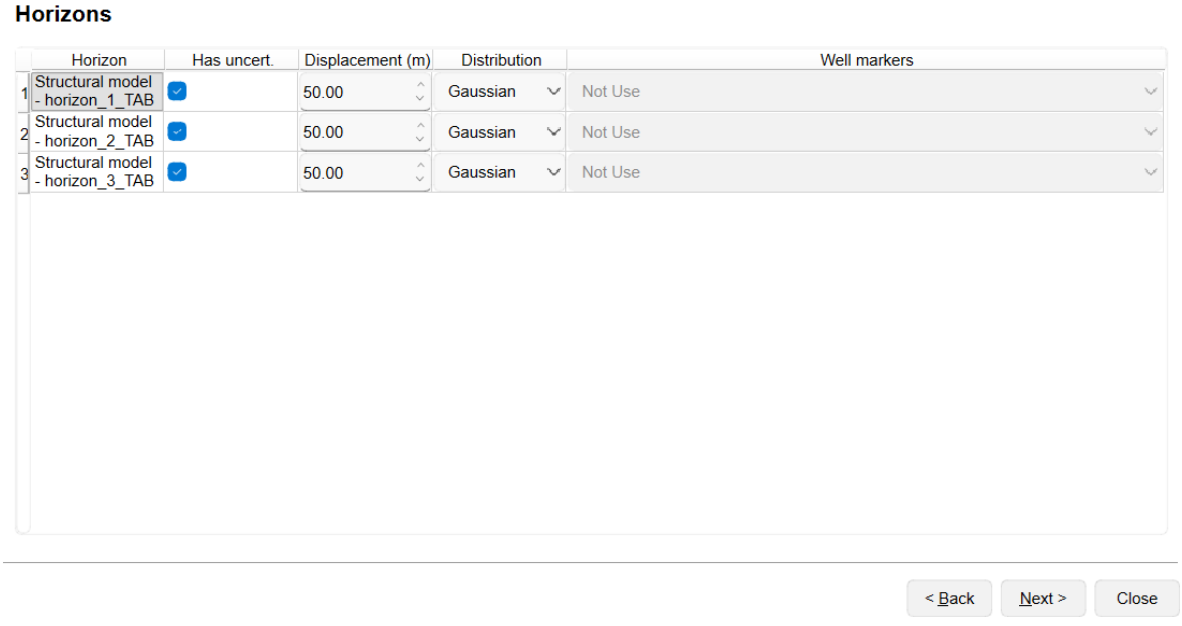
- **Has uncert.** — checkbox, checked by default. Uncheck to fix the fault at its base-case position so it is not perturbed in any realization. When unchecked, the **Displacement** and **Distribution** cells for that row are greyed out.
- **Displacement (m)** — the maximum symmetric displacement applied along the fault-normal direction, in metres. Default 50. The value defines a symmetric envelope of ±D about the base-case fault plane (Constant Symmetric method — the same maximum is applied on both sides of the fault).
- **Distribution** — the probability distribution used to draw the displacement within the envelope. One of **Uniform**, **Triangular**, or **Gaussian** (default).
- **Well markers** — **Use** or **Not Use**. When set to **Use**, the perturbed fault is constrained to stay on the correct side of the well markers defined for it. The column is greyed out and shows **Not Use** when no markers exist for the fault.

Distribution types

- **Uniform** — every position within the envelope is equally likely.
- **Triangular** — positions near the base case are more likely; probability decreases linearly toward the envelope boundary.
- **Gaussian** — positions near the base case are more likely; probability decreases smoothly and is lower at the envelope boundary than Triangular.

Horizons Tab: Horizon Uncertainty

This page lists every horizon in the model, one per row, using the same columns and behavior as the Faults page. Horizons are displaced vertically.



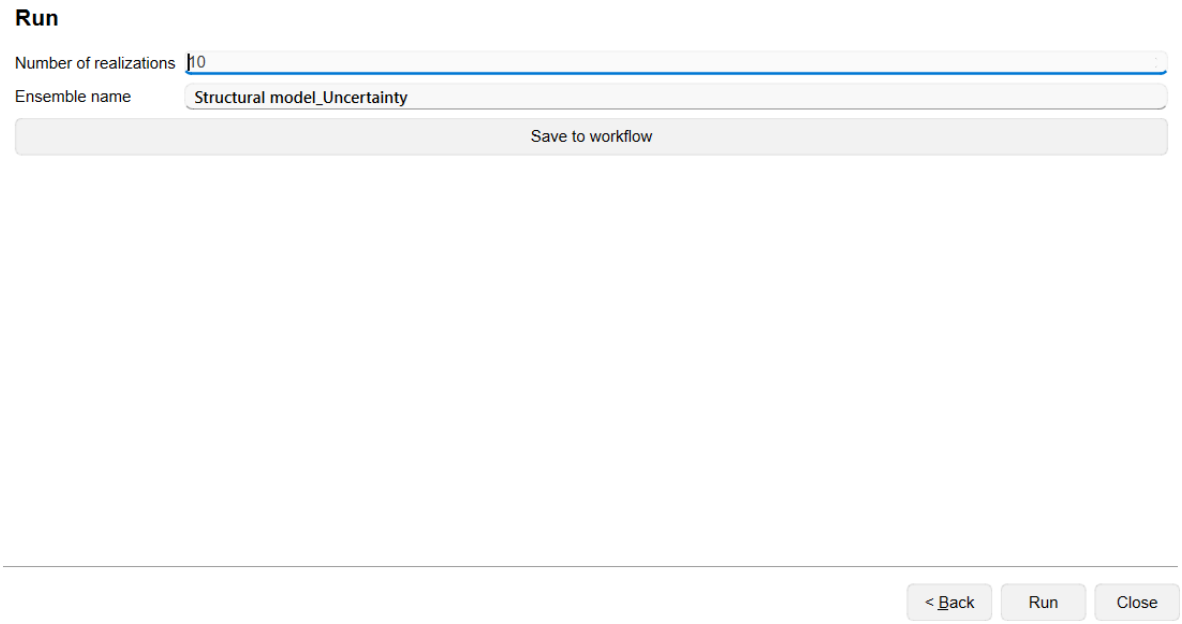
- **Horizon** — read-only name of the horizon.
- **Has uncert.** — checkbox, checked by default. Uncheck to fix the horizon at its base-case position; the **Displacement** and **Distribution** cells are then greyed out.

© 2026 Geomage 2003 LTD

- **Displacement (m)** — the maximum symmetric vertical displacement, in metres. Default 50. Applied uniformly across the horizon extent, symmetrically above and below the base-case surface (Constant Symmetric method).
- **Distribution** — **Uniform**, **Triangular**, or **Gaussian** (default), as described for faults.
- **Well markers** — **Use** (default) or **Not Use**. When set to **Use**, the perturbed horizon is clamped so it stays on its base-case side of the well markers. Greyed out and set to **Not Use** when no markers exist for the horizon.

Run Tab: Run Simulation

The final page sets the ensemble size and name and starts the simulation.



- **Number of realizations** — the number of perturbed models to generate. Default 10 (range 1–1000). Realizations are computed sequentially.
- **Ensemble name** — the name of the output ensemble group. Defaults to `<ModelName>_Uncertainty`.
- **Save to workflow** — instead of running immediately, saves the configured analysis as a step in the current workflow so it can be replayed later.

Click **Run** to start the simulation, or **Close** to cancel without generating an ensemble.

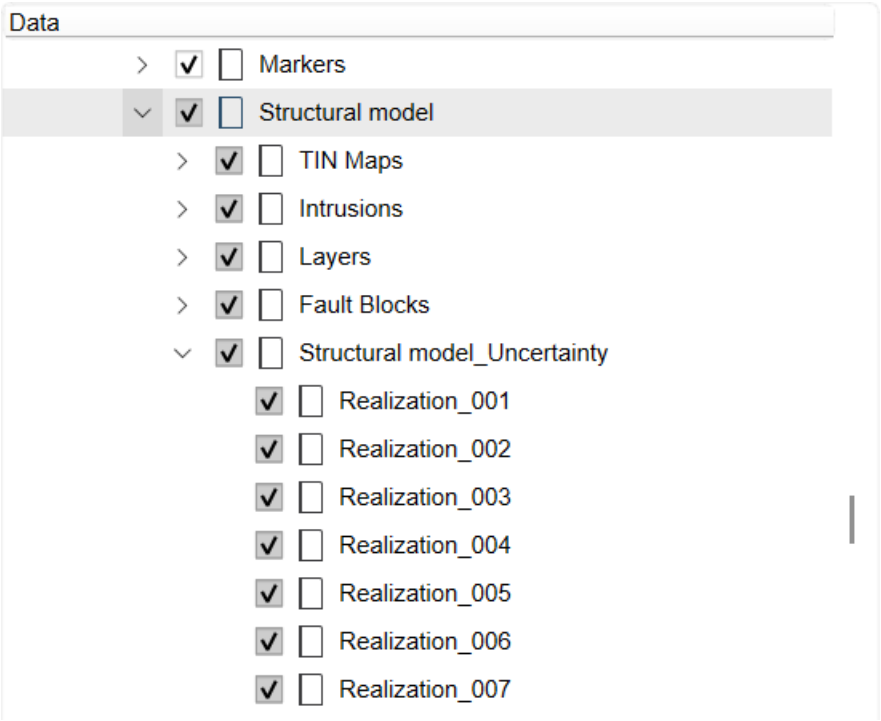
Running the Analysis and Results

When you press **Run**, the wizard generates the requested number of realizations one after another, drawing an independent displacement for each element that has **Has uncert.** checked.

© 2026 Geomage 2003 LTD

Elements with **Has uncert.** unchecked remain at their base-case position in every realization. A progress dialog reports the number of completed realizations.
All realizations are stored together as a **Structural Model Ensemble** group under the base model in the Data Manager:

- The ensemble folder is named `<ModelName>_Uncertainty`. The object properties of the base model report the **Uncertainty ensemble** name and the number of **Realizations**; the properties of a single realization also report its **Realization** index and its **Base model**.
- Each realization (`Realization_001`, `Realization_002`, ...) is a single leaf item with no sub-items; toggling its visibility shows all of its perturbed faults and horizons together.
- Individual realizations can be extracted, renamed, or deleted from their context menu.



Visualizing the Ensemble

- **3D Viewer** — toggling the ensemble container visible renders all realization surfaces simultaneously, color-coded by realization index, producing an uncertainty envelope view. Individual realizations can be toggled on or off, and the base-case model can be displayed alongside for reference.
- **Cross-sections** — all realization horizon and fault traces are drawn per section, color-coded by realization index, while the base-case traces keep their own surface colors.

Recommendations

© 2026 Geomage 2003 LTD

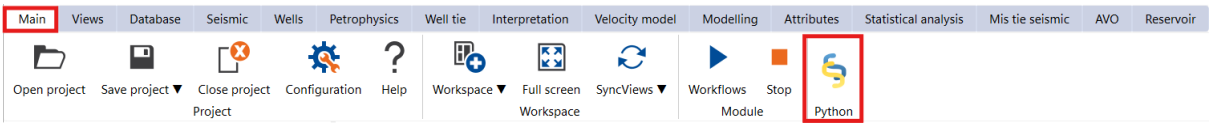
- Start with a small number of realizations (for example the default 10) to validate the setup, then increase it for a fuller uncertainty envelope.
- Uncheck **Has uncert.** for well-constrained elements you want to hold fixed, so the analysis focuses on the poorly constrained surfaces.
- Set **Well markers** to **Use** for elements tied to wells so realizations stay geologically consistent with marker data.
- Use **Gaussian** or **Triangular** distributions when the base case is the most likely position; use **Uniform** when any position within the envelope is equally plausible.
- Enable **Horizontal displacement only** when fault throw is well known but lateral position is uncertain.

See Also

- Structural modeling 798
- Structural models 163
- Conceptual model 191
- Geological modeling 758
- Fault model 181

9.20 Python scripting

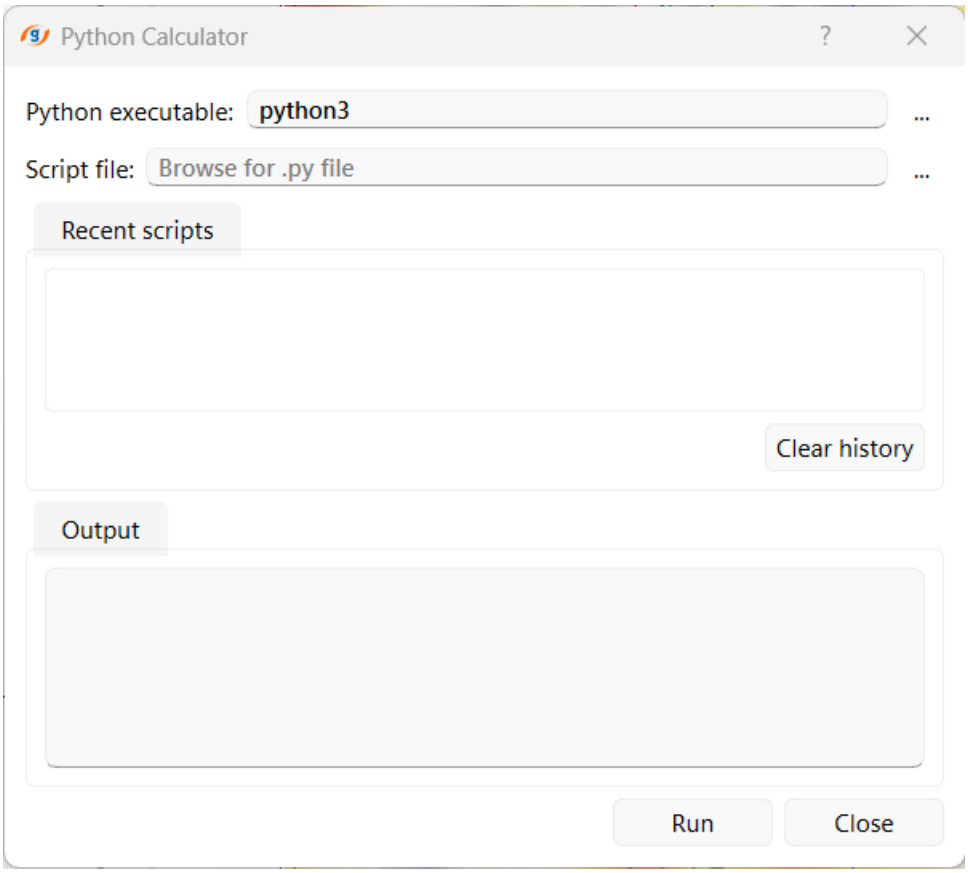
The **Python scripting** plugin lets users run Python scripts inside g-Space to extend built-in procedures, automate repetitive tasks and prototype custom calculations. Scripts can be run from the **Python Calculator** or directly with the **Run Python script** action. The calculator captures stdout, stderr and any exception and displays them in its own **Output** pane, rather than the separate Output View 219 panel.



Running from the Python Calculator

1. On the **Main** tab, click **Python calculator** in the **Python** section.
2. Provide the path to the Python interpreter in **Python executable**. Default: `python3`.
3. Browse to the target script in **Script file**.
4. Optionally, reload a previously used script by double-clicking it in the **Recent scripts** list (the last 20 scripts are remembered). Use **Clear history** to empty this list.
5. Click **Run**. The wizard remains open so scripts can be re-run repeatedly while tuning parameters.

© 2026 Geomage 2003 LTD



Running from Module Properties

To run a script without opening the calculator, set **Python executable** and **Script file** under **Python scripting** in Module Properties 197, then choose **Run Python script**.

Execution model

- Scripts run in a subprocess spawned from g-Space. The wizard captures the full stdout and stderr and prints them to its embedded Output pane once the subprocess finishes.
- If the script raises an unhandled exception the traceback is routed to the embedded Output pane and the wizard reports the failure without disturbing the project.
- Any file the script produces on disk can then be imported back into the project via the usual import wizards.

Accessing project data

© 2026 Geomage 2003 LTD

A script can read and write g-Space project data directly through the built-in `gspace` Python module, so data does not have to be exported to disk first.

This requires Python 3.8 or newer with `numpy` installed on the machine that runs the script.

While a script runs, g-Space opens a local TCP server on `127.0.0.1` (a random port) and the subprocess connects back to it. The script uses the `gspace` module to request data and to write results back: metadata travels as JSON and large numerical arrays as raw binary `numpy` arrays, while the script's own `print()` output continues to appear in the Output pane. Every object handed to the script is a *copy*, so a script can never corrupt project data held in memory.

Data is reached through `gspace.project`, which exposes the project collections, including:

- Wells, horizons and faults.
- Seismic, velocity models and grids.
- Maps, polygons and wavelets.

A minimal script imports the module and works through `gspace.project`. For example, to shift a horizon and write it back:

```
import gspace
h = gspace.project.horizons[0]
points = h.as_numpy() # copy of the horizon points
points[:, 2] += 10.0 # shift Z
h.set_points(points) # write back
```

The script runs as a task, so it can be cancelled while it works. Long-running scripts should call `gspace.check_cancelled()` regularly inside their loops so the **Stop** action is detected promptly; a script that does not stop within a few seconds is terminated. Progress can be reported back to g-Space with `gspace.progress`.

When to use

- Automating data preparation steps that are tedious to drive through the GUI.
- Running third-party geophysical libraries on project data and importing the result.
- Prototyping a calculation before turning it into a native procedure.
- Reading or modifying project data (wells, horizons, seismic and more) programmatically through the `gspace` module.

See Also

- Workflows 535

© 2026 Geomage 2003 LTD

- Output View 219

9.21 Well Data QC and Verification

After every well data import, verify the result before you continue. **Quality control (QC)** catches mismatched columns, wrong units, misplaced wells and unrealistic time-depth relationships while they are still easy to fix. The *Wells data workspace* gathers every view you need for this check in a single layout.

Open the QC Workspace

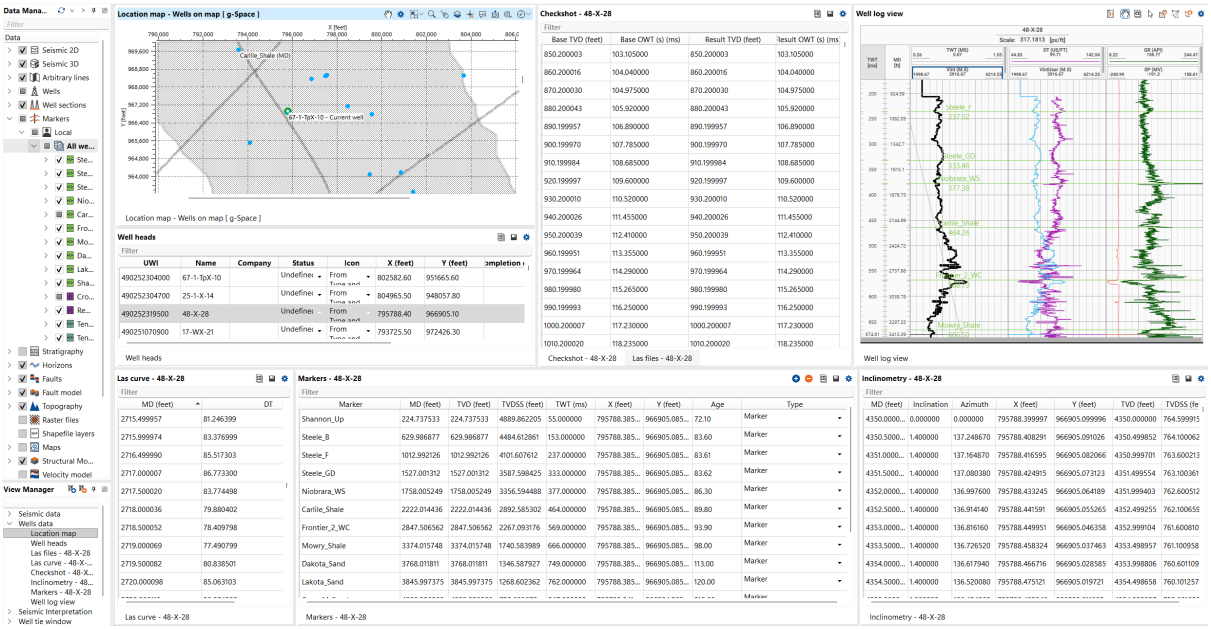
Select the **Wells** data workspace in the **View Manager** (or click **Views** on the Wells bar) to open the pre-configured layout: a Location map, the Well heads table, the Inclinometry view, the Well log view, the LAS files and LAS curve tables, and the Checkshot table. Each table refreshes for the well you select on the map, so you can click through wells and QC them one by one. See Wells data workspace 232 for the full layout.

Verification Checklist

Work through the following checks after each import:

1. **Location map check** — confirm wellhead positions, trajectories and marker locations fall where you expect on the map.
2. **Well heads table QC** — review the coordinates, Kelly bushing (KB), total depth, status and other header attributes in the Well heads table.
3. **Trajectory validation** — check the well path on the map and in the Well log view; verify the deviation and depth reference look reasonable.
4. **Log curve review** — inspect the loaded curves in the LAS files table; review units, curve types, histograms and value ranges for outliers or null spikes.
5. **Marker consistency** — check marker positions in the Well log view and confirm they display correctly on the MD, TVD or TWT axis.
6. **Checkshot verification** — review the checkshot data in the table and the Well log view; confirm the time-depth relationship is monotonic and realistic.

© 2026 Geomage 2003 LTD



Recommendations

- Click a well on the Location map to refresh every table at once, then scan the tables for that well before moving on.
- Copy table values with the **Copy** tool and paste them into a spreadsheet for side-by-side comparison with the source files.
- Edit obvious mistakes in place — editable fields such as checkshot values are changed by double-clicking the cell.
- Re-run the relevant import wizard if columns were mapped incorrectly rather than patching values one by one.

See Also

- Wells data workspace 232
- Import well headers 393
- Import LAS files 407
- Import markers 403
- Import checkshots 414
- Well correlation view 296

© 2026 Geomage 2003 LTD

- D -

Displaying 3D volume 244

- G -

Gather conditioning (including NMO, RMO and parabolic Radon) 10

- I -

Import prestack gathers 2D/3D 10

- P -

Polynomial analysis of AVO 10

- R -

Ribbon bar 235

© 2026 Geomage 2003 LTD