

**INSTALLING AND RUNNING THE  
THREE-DIMENSIONAL TOMOGRAPHY PROGRAM**

**GEOTOMCG**

**This is the demo version of GeoTomCG.**

The only data files that the demo version can read are the data files included with the program set. Trying to read any other data files will make the program exit.

**GeoTom, LLC  
13785 Diamond Path  
Apple Valley, MN 55124**

May 2015

# INSTALLING AND RUNNING GEOTOMCG

## CONTENTS

	<u>Page</u>
Acknowledgements and Request For Suggestions	4
New Features and Options	4
Quick Start	5
Comienzo Rápido (Quick Start in Spanish)	8
Overview of GeoTomCG	11
3D Grid of Points	12
Curved Ray Tracing	12
Inversion Method	12
Previous Programs	13
Installing and Starting GeoTomCG	14
Generating and Editing Input Files	15
Data Files (Source and Receiver Coordinates, Traveltimes)	15
Synthetic Data File	15
Data in Standard 3D Format, One Measured Variable	17
Data in Combination 3D Format, Two Measured Variables	18
Data in 2D and 3D TOM Format	18
Data in Alternative 2D Format	18
Plotting Variables in Data Files	18
Removing Outliers in Data Files	22
Rotating Coordinates	23
Initial Velocity Model	24
Generating the Model	24
Specifying Anisotropy	25
Saving the Model	26
Editing the Model	27
Drawing on Grid and Applying Horizontal Layers	28
Applying 3D Tilted Layers	29
Specifying Constraints and Regularization	35
Point by Point	35
Entire Grid	36
Reflecting Plane (Only For Reflected Rays)	37
Performing Inversion Calculations	38
Open Data File	38
Open Initial Velocity Model (Grid-point) File	38
Select Inversion Options	38
Iteration and Velocity Limits	38
Smoothing	39
Damping	40
Anisotropy Type	40

	<u>Page</u>
Specify Raypath File Name (Optional)	41
Specify Reflector File (Only For Reflected Rays)	41
Start Inversion	41
Output Files Generated Automatically	42
Calculated Velocity Model File	42
Residuals File	42
Raypath File	42
Displaying Tomograms and Saving Output Files	43
Copying to Windows Clipboard	48
Slice File, Slice Aligned with Coordinate Axes	49
Diagonal Slice File	49
Exported Files for 3D Graphics Programs	50
Tecplot or Voxler Format	50
Slicer Dicer Format	51
Exported 2D Raypath File for Surfer	51
Plots and Files from Residuals Files	52
Plotting Variables	52
File of Calculated Times in Data File Format	53
File of Measured Times with Outliers Removed	53
Checking For and Applying Anisotropy	54
Example 1. No anisotropy	55
Example 2. Ambiguous evidence of anisotropy, unconfirmed by residuals	56
Example 3. Anisotropy with the maximum velocity in the horizontal plane.	57
Example 4. Anisotropy with the maximum velocity in a dipping plane.	58
Example 5. Nonuniform anisotropy and editing the velocity model accordingly.	60
Comparing Velocity Models	61
Reading a Series of Data Files Automatically	61
Special Procedures for Attenuation Tomography	62
Ray Tracing in Amplitude Inversions	62
Interpretation of Attenuation Tomograms	62
What is Amplitude?	63
Elastic Wave Amplitudes	63
Electromagnetic Wave Amplitudes	64
Radiation	64
What is Attenuation?	64
Geometric spreading	64
Absorption (intrinsic attenuation)	65
Scattering	65
Quantifying Amplitudes	65
Amplitude Data Reduction	67

	<u>Page</u>
Illustrated Examples of Amplitude Processing	70
Example 1, Spherical Source Emission	70
Example 2, Nonspherical Source Emission	71
Example 3, Nonspherical Source Emission, Amplitude in dB	72
Demo Data Sets and Tutorial	74
Wavefront Construction	77
Use of wavefront construction in GeoTomCG	77
Reason this option is offered	77
How to use the wavefront construction option	78
Method of Wavefront Construction in GeoTomCG	79
Initiation and Propagation	80
Arrivals of wavefronts at receivers	82
References	84
Index	86

## **ACKNOWLEDGEMENTS AND REQUEST FOR SUGGESTIONS**

Dr. Thomas Fechner, Geotomographie, Neuwied, Germany suggested the option for rotating coordinates in this version and has contributed several suggestions implemented in past versions. Dr. Fechner, Lutz Karl, Markus Kunert (intern) and Wendy Albers (intern) of Geotomographie were instrumental in the development of the quality parameters in this version.

Larry Olson, PE, President and Principal Engineer, Olson Engineering, Inc., Wheat Ridge, CO provided suggestions that resulted in many of the features in the 3D displays, including special features for processing data from concrete pilings and several more general suggestions.

The author welcomes all suggestions for improving the software and the documentation. Please send any suggestions to sales@geotom.net. The author has tested the new features in this version, but a user might encounter a problem with other data or other combinations of options. Please report any problems immediately.

## **RECENT FEATURES AND OPTIONS**

1. The newest set of editing options for velocity model files allows the user to apply gradients, layering, or anisotropy between parallel planes tilted in any direction except vertical.
2. Generating synthetic data files with uniform velocity and forward modeling with a user-specified velocity pattern were simplified.
3. The 3D rotatable display of ray direction vs. ray straight-line velocity was improved.
4. Data quality and Slowness residual/slowness were added to the display of calculated parameters at grid points. Data quality is a combination of the data quality weighting factors for raypaths affecting the calculated velocity at each grid point. Slowness residual/slowness indicates the goodness of fit for raypaths affecting the calculated velocity at each grid point.
5. Optional quality weighting factors can be added to the data files. They multiply correction factors when fitting data, so they should be 1 for good data and less than 1 for poorer data.
6. Several additional data input options were added.
7. X and Y coordinates in the data file can be rotated about the Z axis.
8. Coordinates are double-precision floating point numbers, so 10-digit GPS coordinates in the form 1234567.123 can be used. GPS coordinates in other forms must be converted to that form.
9. The user can select 3D rectangle and 3D cylinder views of the velocity display, in addition to the three 2D displays available in prior versions. The 3D views can be rotated and tilted.
10. The user can select a color spectrum with red indicating slower and green faster velocities. This spectrum was requested for concrete work to use red to magenta as an indicator of low-velocity possible problem zones and green as zones without problems, with yellow between green and red. The traditional red = fast and blue = slow spectrum is still available.

## QUICK START

A user who wants to start running GeoTomCG without reading much of the manual can begin by following the instructions in this section. Only the most commonly used options are described.

If GeoTomCG is on a CD, run SetupGeotomCG.exe in the GeoTomCG subdirectory to start the installation. (For versions of Windows before Win2000, please contact the author by e-mail at sales@geotom.net to ask for an alternative installation program.) Then follow the on-screen instructions. The installation program will install the GeoTomCG program and sample data files in the chosen subdirectory, and will add GeoTomCG to the Windows list of programs. If installation fails, send an e-mail to sales@geotom.net. Sample files include a standard 3D (\*.3dd) data file, an initial velocity model file (\*.3mq), and other less used files. If sent by e-mail, the files are in a password-protected compressed zip file, called GeoTomCG(version).zip. The password is "geotom". Some virus protection programs block compressed files containing an executable file unless they are protected by a password. Unzip GeoTomCG(version).zip to a subdirectory on your hard disk, then run SetupGeoTomCG.exe and follow the on-screen instructions.

Start GeoTomCG by clicking on the name in the Windows program list or switching to the GeoTomCG subdirectory and double-clicking on the program name GeoTomCG.exe. The program will start and display a Windows program menu.

If you have prepared a data file with the time-picking program TomTime, read your data file by clicking **Data, Open data file**, and select it in the file selection dialog box. The first 1000 lines of a data file are shown to allow you to verify it is the correct file and was read correctly. If you have not made a data file, you can use the sample Demo1.3dd as a guide in preparing one. The data file format is as follows:

Two lines of general information, up to 80 characters in each line.

One line for each raypath, containing

Ray identifier	Source X, Y, Z	Receiver X, Y, Z	Time
----------------	----------------	------------------	------

Ray identifier is a positive integer. Source and receiver X, Y, Z coordinates and time are floating point numbers. They do not need to be in particular columns, but must be separated by spaces, commas, or tabs. Units are not specified or shown in GeoTomCG, so coordinates and time can be in any units. Calculated velocities are in the combination of units for coordinates and time. For example, Demo1.3dd coordinates are in feet and time in milliseconds, so velocities are in ft/ms.

If desired, display a plot of the data by clicking **Data, Plot**. A dialog box with plotting options will appear in the upper left corner of the screen. The user can select variables for the horizontal and vertical axes of the plot, and can enter new values for their ranges. The plot axes span the ranges in the dialog box, and may be somewhat larger to make the tic marks occur at intervals of 1, 2, or 5. The separate entry for ray identifier range allows the user to select a combination of a ray identifier range and some other variable for the horizontal axis. When the variables and ranges are satisfactory, click **OK** to display the plot.

When the variable on the horizontal axis is the ray identifier, the user can determine the ray identifier for a particular point by right-clicking on or just to the right of the point and, if necessary, using the left and right arrow keys to move the indicator by one unit. The small line at the top shows the approximate position and the caption at the top shows the exact identifier.

To generate an initial velocity model (grid-point) file after reading your data file, click **Model, Generate model**, and enter the parameters requested in the text boxes. Units must be consistent with the data file. Save your generated initial velocity model file. Then load it by clicking **Model, Open model file**, and selecting it in the file selection dialog box. The first 1000 lines of the file will be shown. The file header lines are described in the detailed instructions. The format for the lines in the model file describing grid points is:

X Y Z Velocity Constraint Anisotropy Data quality Slowness residual/slowness

Data quality and Slowness residual/slowness are set to 1 for the initial model. The initial values are not used in the inversion. The final values are calculated during the inversion. A sample velocity model file Demo1.3mq is provided.

Select options for running by clicking **Options, Inversion parameters**. The top two text boxes allow you to enter the number of straight-ray and ray-bending (curved-ray) iterations. Inversions typically take 10 to 15 iterations to obtain stable residuals. The default 1 straight-ray iteration and 7 ray-bending iterations are adequate for quick tests. The next set of text boxes is for entering wavefront iteration parameters. You will usually not use these boxes. The bottom two text boxes allow you to enter minimum and maximum allowed velocities. The default minimum allowed velocity is half of the smallest straight-line velocity for the data file, and the default maximum is double the largest straight-line velocity. Enter the desired values in the text boxes, and click **OK**.

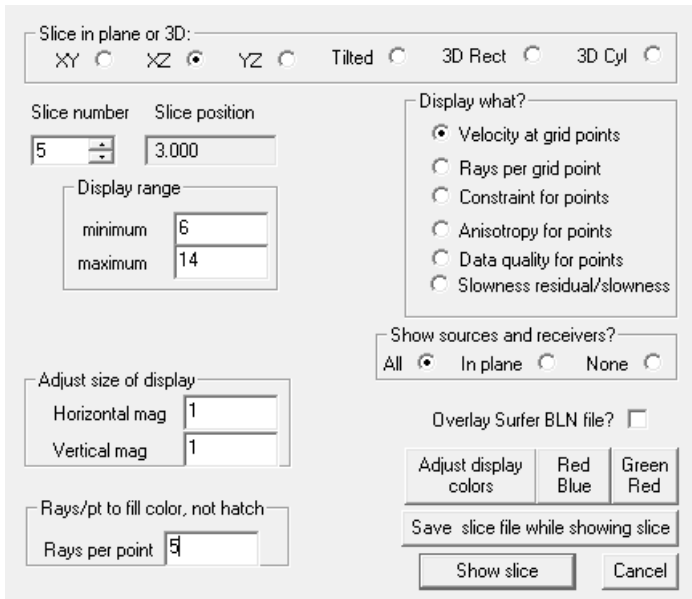
If desired, prepare to save raypaths by clicking **Raypaths, Save Raypaths**, then provide a name for the raypath \*.ray file to be saved. A raypath file contains the 3D coordinates of raypaths that can be read and shown while displaying the model.

Run the inversion by clicking **Run** on the main menu. Two options will be displayed, **Run standard inversion** and **Run curved-ray attenuation**. Click **Run standard inversion**. To indicate progress, the program displays the ray identifier for every hundredth raypath completed and the RMS residual and sum of residuals for each completed iteration. A message box indicates when the program execution is complete.

A velocity model file with default extension 3mq is generated each iteration. It contains the 3D coordinates and calculated velocities for each point in the format of the input velocity model. It is generated each iteration so you can stop execution before the final iteration and have results. The previous file is overwritten each iteration. You are asked to provide a name for the model \*.3mq file after clicking the menu options **Run**, then **Run standard inversion**.

A residuals file with default extension rsd is generated after the final iteration. It contains the RMS (root-mean-square) residual for each iteration and the residual for each raypath for the final iteration. The residual equals measured time minus calculated time. You are asked to provide a name for the residuals \*.rsd file after providing a name for the \*.3mq model file.

A tomogram is a 2D representation of the calculated velocities. GeoTomCG displays tomograms with colors in small rectangles representing the velocity at the grid-point in the center of each rectangle.



After running the program, open the calculated velocity model file by clicking **Model, Open model file**, and selecting the file name in the file selection dialog box. Then click **Model, Display model**. The dialog box shown to the left will appear in the upper left corner of the screen. Select a 2D slice plane or a 3D view with the radio buttons. In 2D views, select the initial slice number with the up-down control, which also indicates the slice position in the plane. Select what parameter to display. Click **Show slice**. The slice will be shown in a scrollable window. The source and receiver positions from the open data file are shown as S and R, respectively, if **All** is selected under **Show sources and receivers?** The

positions are shown for all sources and receivers, not just the ones within the slice being viewed. A title at the top shows the slice number and position. The slice number can be changed with the Page Up and Page Down keys.

Raypaths can be overlaid on the tomogram if a raypath file is read by clicking **Raypaths, Read raypaths**. Grid points with fewer raypaths than the number in the text box above labeled **Rays per point** are shown with a hatched pattern around that point. Rays are shown for all slices, so rays may appear where the hatched pattern indicates no raypaths in that slice.

While raypaths are displayed, right-clicking on a source position limits the displayed rays to those from that source. Then left-clicking on a receiver position will show the ray identification and measured and calculated times. To resume displaying all rays, click **Raypaths, Resume raypath display**. The cursor must be no more than  $\frac{1}{4}$  of a voxel width or height away from the source or receiver position to select it.

The size of the display in the horizontal and vertical directions can be adjusted by clicking on **Model, Display Model**, then entering new values for the horizontal and vertical magnification factors in the lower left text boxes. This feature can also provide vertical or horizontal exaggeration by using different values for the horizontal and vertical magnification factors.

The variables in residuals (\*.rsd) files can be plotted. Open a residuals file by clicking **Residuals, Open residuals file**. Then click **Residuals, Plot**, and a dialog box with plotting options will appear in the upper left corner of the screen. The option for plotting ray angle versus velocity is to help investigate possible anisotropy. Ray angle is calculated from the difference between the Z coordinates for source and receiver divided by the source-receiver separation. Normalized residual is the residual divided by the measured time. When the variable on the horizontal axis is the ray identifier, the user can determine the ray identifier for a particular point in the same manner as in the data plot.



## COMIENZO RÁPIDO (Quick Start in Spanish)

Translation to Spanish courtesy of José Carlos Cambero Calzada,  
Senior Geophysicist, Hager-Richter Geoscience, Inc

Un usuario que desee comenzar a usar GeoTomCG sin la leer todo el manual puede comenzar por leer las instrucciones que incluimos a continuación donde solamente se describen las opciones más comúnmente usadas.

Si el programa GeoTomCG está en un CD, active SetupGeotomCG.exe en el subdirectorío GeoTomCG para comenzar la instalación. (Para pre-Win2000, envíe un E-mail a sales@geotom.net.) Entonces siga las instrucciones en pantalla. El programa instalará GeoTomCG y varios archivos de ejemplo en el subdirectorío elegido, y agregará GeoTomCG a la lista de programas de Windows. Si la instalación falla, envíe un E-mail a sales@geotom.net. Los archivos de ejemplo incluyen un fichero de datos estándar 3D \*.3dd, un archivo del modelo de nodos de malla (\* 3mq), y otros archivos menos usados. Si son enviados por E-mail, los archivos están en un archivo comprimido protegido con una contraseña, llamada GeoTomCG(version).zip. La contraseña es "geotom". Ciertos programas antivirus bloquean archivos comprimidos a menos que sean protegidos por una contraseña. Descomprima GeoTomCG(version).zip en un subdirectorío de su disco duro. Después de descomprimir GeoTomCG(version).zip, active SetupGeoTomCG.exe y siga las instrucciones en pantalla.

Para comenzar GeoTomCG, clic el nombre en la lista del programa de Windows o cambiando al subdirectorío de GeoTomCG y haciendo doble clic en el nombre GeoTomCG.exe. El programa comenzará y exhibirá un menú típico de Windows.

Cargue el fichero de datos de muestra Demo1.3dd seleccionando **Data, Open data file**, y seleccione Demo1.3dd en el cuadro de diálogo de la selección del archivo. Aparecerán las primeras 1000 líneas del fichero de datos para permitir que verifique que el archivo es correcto. Usted puede utilizar este fichero de datos como guía en la preparación de sus propios archivos. El formato de archivo es como sigue: Dos líneas de información de carácter general, hasta 80 caracteres en cada línea. Una línea para cada trayectoria de onda, conteniendo

Identificador	Fuente X, Y, Z	Receptor X, Y, Z	Tiempo
---------------	----------------	------------------	--------

Identificador de trayectoria de onda es un número entero positivo. Coordenadas X,Y,Z de la fuente y el receptor y el tiempo son números de coma flotante. Estos números no necesitan estar en columnas diferenciadas, se deben separar por espacios, comas, o tabuladores. Los coordenadas y el tiempo pueden estar en cualquier unidad conveniente. Las velocidades calculadas estarán en cualquier combinación de unidades utilizada para coordenadas y tiempo. Los resultados no muestran etiquetas de unidades.

Si lo desea, puede obtener un gráfico de los datos seleccionando **Data, Plot**. Un cuadro de diálogo con varias opciones aparecerá en la esquina superior izquierda de la pantalla. El usuario puede seleccionar las variables para los ejes horizontales y verticales del diagrama, y puede cambiar los valores de sus rangos. El rango de los ejes del diagrama excede los valores máximo y mínimo para que aparezcan marcas a 1,2, y 5. La entrada por separado del identificador de trayectoria de onda permite que el usuario seleccione una combinación del rango del identificador y de otra variable para el eje horizontal. Cuando las variables y los rangos son satisfactorios, seleccione **OK** para obtener el gráfico.

Cuando la variable en el eje horizontal es el identificador de trayectoria de onda, el usuario puede determinar el identificador para un punto en particular colocando el cursor a la derecha del punto o haciendo clic sobre cada punto, y en caso de necesidad, usando las flechas izquierda y derecha para

mover el indicador por el diagrama. La línea pequeña en la parte superior muestra la posición aproximada y el texto en la parte superior muestra el identificador exacto.

Cargue el archivo ejemplo del modelo en forma de nodos de la malla Demo1.3mq haciendo clic en **Model, Open model file with quality parameters**, y seleccionando Demo1.3mq en el cuadro de diálogo de la selección del archivo. Las primeras 1000 líneas del archivo serán mostradas. Usted puede generar sus propios archivos modelo seleccionando **Model, Generate model**, e incorporando los parámetros solicitados a los cuadros de texto. Las unidades deben ser consistentes con el fichero de datos. Para una descripción de las líneas del cabecera en el archivo modelo vea las instrucciones detalladas. El formato para las líneas en el archivo modelo que describe los nodos de la malla es:  
 X Y Z Velocidad inicial Constreñimiento Anisotropía Datos calidad Lentitud residual/lentitud

Seleccione las opciones de inversión haciendo clic en **Options, Inversion parameters**. Los dos cuadros de diálogo superiores permiten que usted seleccione el número de iteraciones para rayo-recto y para rayo-curvo. Las inversiones finalizan normalmente tras 10 a 15 iteraciones para obtener residuales estables. Por defecto 1 iteraciones de rayo-recto y 7 iteraciones de rayo-curvo son adecuadas para las pruebas rápidas. El sistema siguiente de cuadros de diálogo se utiliza para modificar los parámetros de la iteración del frente de onda. Estas opciones no se utilizan normalmente. Las dos opciones inferiores permiten que modifique el mínimo y máximo de las velocidades permitidas. El mínimo por defecto es la mitad de la velocidad directa más baja y el máximo por defecto es el doble de la velocidad directa más alta. Incorpore los valores deseados , y seleccione **OK**.

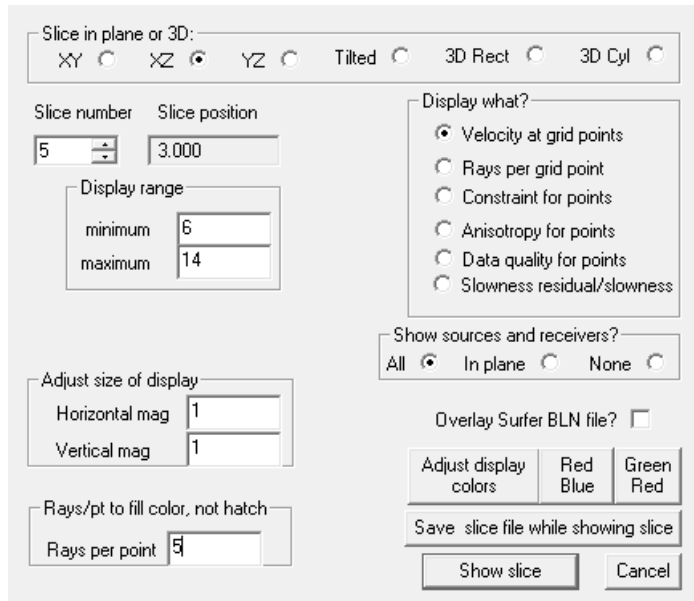
Si lo desea puede archivar los trayectoria de ondas seleccionando **Raypaths, Save Raypaths**, y proporcionando el nombre \*.ray que desea guardar. El archivo contiene los coordenadas 3D de los trenes de ondas que pueden ser leídos y mostrados mientras se dibuja el modelo.

Comience la inversión seleccionando **Run** en el menú principal. Aparecerán dos opciones, **Run standard inversion** y **Run curved-ray attenuation** . Seleccione **Run standard inversion**. Como indicador de que el programa funciona, aparecerá el identificador de trayectoria de ondas cada 100 datos procesados, la residual RMS y la suma de las residuales para cada iteración terminada. Un cuadro de mensaje indica cuando el cálculo ha finalizado.

En cada iteración se genera un archivo modelo con la extensión los 3mq. Contiene los coordenadas 3D y las velocidades calculadas para cada nodo con las unidades de la velocidad de partida. Si lo desea puede parar la ejecución antes de la iteración final y tener resultados. El archivo anterior se sobre escribe en cada iteración. Se le pedirá un nombre para el archivo del modelo \*.3mq después de seleccionar **Run, Run standard inversion**.

Las residuales se archivan con la extensión .rsd y se generan después de la iteración final. Contiene la residual RMS para cada iteración y la residual para cada tren de ondas de la iteración final. La residual es igual a tiempo medido menos tiempo calculado. Se le pedirá un nombre para los archivos de residuales \*.rsd.

Un tomograma es una representación en 2D de las velocidades calculadas. GeoTomCG proporciona tomogramas en forma de pequeños rectángulos coloreados que representan la velocidad en el nodo en el centro de cada rectángulo.



Después de hacer funcionar el programa, cargue el archivo del modelo de velocidad calculada seleccionando en **Model, Open model file with quality parameters**. Entonces seleccione **Model, Display model**. El cuadro de diálogo aparecerá en la esquina izquierda superior de la pantalla. Seleccione el plano o rebanada que quiera representar. Seleccione el número del plano con el control Page up-down, que también indica la posición de cada plano. Seleccione qué parámetro representar haciendo clic en **Show slice**. El plano aparecerá en una ventana aparte. Las posiciones de la fuente y receptor del fichero de datos abierto aparecerán en el gráfico como S y R, respectivamente, si selecciona **All** debajo de **Show sources and receivers?**. Las

posiciones se muestran para todas las fuentes y receptores, no solo las que están dentro del plano representado. Un texto en la parte superior muestra el número y la posición del plano. El número del plano se puede cambiar con el control Page Up-down.

Los trayectoria de ondas pueden ser superpuestos si se abre un archivo seleccionando **Raypaths, Read raypaths**. Los nodos de la malla con menos trenes de ondas que el número que aparece en el cuadro de mensaje etiquetado como **Rays per point** se muestran con un tramado alrededor de ese punto. Los rayos se muestran para todas las rebanadas, por lo que los rayos pueden aparecer donde el patrón tramado no indica ningún trayectoria de ondas en esa rebanada.

Cuando tiene en pantalla los trayectoria de ondas, si selecciona con el botón derecho del ratón sobre la una posición de la fuente aparecerán solo los rayos relacionados con esa fuente. Si hace clic el botón izquierdo del ratón sobre un receptor, aparecerán los identificadores de rayo y el tiempo medido y calculado. Para mostrar todos los rayos, seleccione **Raypaths, Resume raypath display**. El cursor no debe ser más que  $\frac{1}{4}$  de la anchura o altura del voxel inmediatamente después de la posición de la fuente o del receptor para que pueda ser seleccionado.

El tamaño del gráfico en las direcciones horizontal y vertical puede ser ajustado seleccionando **Model, Display Model**, y cambiando los valores de factor de escala o de magnificación deseados para los ejes horizontal y vertical.

Las variables en archivos de las residuales (\*.rsd) pueden ser representadas. Abra los archivos seleccionando **Residuals, Open residuals file**. Seleccione **Residuals, Plot**, y aparecerá un cuadro de diálogo con varias opciones en la esquina superior izquierda de la pantalla. La opción para representar ángulo del rayo respecto a velocidad es útil para detectar posible anisotropía. El ángulo del rayo se calcula por la diferencia entre las coordenadas Z de la fuente y el receptor dividido por la separación del fuente-receptor. La residual normalizada es la residual dividida por el tiempo observado. Cuando la variable en el eje horizontal es el identificador de trayectoria de ondas, el usuario puede determinar el identificador para un punto particular de manera semejante como en el gráfico de los datos.

## OVERVIEW OF GEOTOMCG

GeoTomCG performs 3D tomographic analyses. Source-to-receiver traveltimes can be analyzed to calculate velocities, or amplitudes can be analyzed to calculate attenuation coefficients. Source and receiver positions can be in any configuration within a 3D grid. The tomographic analysis calculates velocity or 1/attenuation at points within the grid. Anisotropy can be specified for each point of the grid. Raypaths can be straight or curved. The ability to perform 3D analyses is an important advantage in counteracting the problem of nonuniqueness with crosshole data, as intersecting planes can be processed simultaneously.

Site information can be used to constrain the inversion and obtain more reliable results.

Options include:

1. Maximum and minimum calculated velocities.
2. Constant velocity in user-selected areas or volumes in the grid.
3. Groups of points with similar velocities, such as horizontal layers or vertical columns.
4. Constraints that take into account the uncertainty in site knowledge.

GeoTomCG provides graphical displays of velocities, raypath densities, and other parameters for inspecting results and preparing preliminary reports. These displays can be copied to the Windows clipboard and pasted into documents. Plots for quality control include traveltimes versus distance, residual versus raypath angle, velocity versus raypath angle, and others. It also provides 3D numerical results and 2D slices of the 3D grid as ASCII files that can be read by commercial 3D and 2D graphics programs for preparing formal reports and presentations.

Raypaths can be saved in a file and displayed. Curved-ray amplitude inversions can be performed with curved raypaths from prior traveltimes inversions. (Amplitude inversions cannot provide appropriate curved rays because ray bending depends on contrasts in velocity, not attenuation.)

### 3D Grid of Points

The velocity or attenuation model is defined on a 3D grid with rectangular coordinates. The point spacing can vary with direction. Values (velocity or 1/attenuation coefficient) are specified at the points, and computed in the intervening cells (or voxels) by linear interpolation. The grid is oriented with its edges parallel to the coordinate axes defined by the data.

After reading a data file and while generating a model grid of points, GeoTomCG suggests a default grid based on the number of data points and the range of source and receiver locations. The minimum and maximum values of X, Y and Z coordinates in the data file define the limits of the survey volume. To accommodate raypath curvature outside the survey volume, the default grid is larger than the survey volume. If the data are 2D (maximum and minimum source and receiver coordinates in one direction are identical) then the number of points in each of the other two directions is proportional to the source-receiver spread in that direction times the square root of the number of measured traveltimes. If the data are 3D, the number of points in each direction is proportional to the source-receiver spread in that direction times the cube root of the number

of traveltimes. The user can usually select a more appropriate number of points by considering the data. Having fewer points than traveltimes does not guarantee a unique solution.

### **Curved Ray Tracing**

Ray tracing and travel-time computation may be performed with either straight or curved raypaths. The straight-ray approximation allows rapid calculation of model travel times, but its validity diminishes with large velocity contrasts. Curved-ray calculations are slower, but are more accurate for strong contrasts.

Curved ray tracing is performed with a revised form of ray bending, derived from the Um and Thurber (1987) method but modified to give more reliable results where the original Um and Thurber method failed. The ray-bending approach is based on repeated subdivision of an initially straight raypath into an increasing number of straight segments. Each step in the process consists of a division phase and an adjustment phase.

For the initial subdivision step, a straight raypath is constructed from source to receiver, and the midpoint is calculated, dividing the original path into two segments. The velocity gradient is computed at the midpoint, and the midpoint is displaced according to the local gradient. The gradient at the new midpoint is then calculated, and the midpoint location is again adjusted. This adjustment is repeated one or more times before proceeding to the next subdivision step.

In the second subdivision step, the same process is applied to each of the segments produced in the first step. Subsequent subdivision steps proceed in the same manner, with the number of segments doubling each time, producing more smoothly curved paths. At each step, the traveltimes along the path is calculated. Subdividing terminates when a stable minimum time is attained. In the original formulation, the algorithm calculated adjustments assuming the velocity gradient was constant. This assumption led to failures near abrupt changes in the velocity gradient. GeoTomCG tries a range of adjustments to find the one that gives the shortest time path. This revision slows execution, but greatly improves the reliability of results.

### **Inversion Method**

GeoTomCG performs inversions with the simultaneous iterative reconstruction technique, or SIRT (Lytle et al., 1978; Peterson, et al., 1985; Lehmann, 2007, pp. 32-36). SIRT calculations modify an initial velocity model by repeated cycles of three steps: forward computation of model travel times, calculation of residuals, and application of velocity corrections. The cycle repeats through a specified number of iterations. In GeoTomCG, the weighting of velocity correction factors for grid-points is proportional to the sum of the fractions of raypaths in voxels that share that point. The alternative of weighting proportional to the length of raypaths in surrounding voxels gives too much influence to long slanted raypaths.

Lehmann (Lehmann, 2007, pp. 28-32) compared various methods of inversion including SIRT, algebraic reconstruction technique (ART), conjugate gradient (CG) and least square (LSQ) methods. He concluded that SIRT is the best method for field data because "The SIRT algorithm obtains the best and most stable results. With this algorithm, the pictures have a

smoother appearance than others and noise does not critically deteriorate the reconstructed images." He stated that LSQR and CG "work well for noiseless data, but with noise contamination, the results are critically degraded."

## Previous Programs

GeoTomCG is the sixth generation tomographic program in this series. The first 3D program in the series was 3DTOM, released by the U.S. Bureau of Mines shortly before it closed. Part of the discussion of methods is taken from the Bureau of Mines Report of Investigations 9617. That program and the three that preceded it are available for downloading from a USGS web site. The available programs and addresses are as follows:

BOMTOM (1988, RI 9159. 2D straight ray tracing only, but very fast, in FORTRAN)

<http://greenwood.cr.usgs.gov/pub/open-file-reports/ofr-00-0455/>

BOMCRATR (1992, RI 9411. 2D curved ray tracing with shooting rays, in FORTRAN)

<http://greenwood.cr.usgs.gov/pub/open-file-reports/ofr-00-0454/>

MIGRATOM (1994, RI 9497. 2D curved ray tracing with wavefront construction, in PASCAL)

<http://greenwood.cr.usgs.gov/pub/open-file-reports/ofr-00-0457/>

3DTOM (1996, RI 9617. 3D curved ray tracing with ray bending, in PASCAL)

<http://greenwood.cr.usgs.gov/pub/open-file-reports/ofr-00-0456/>

These programs are not recommended for professional use, but the manuals provide informative discussions and the programs may be useful for students or others who want to try various tomographic methods. The most relevant to GeoTomCG is the manual for 3DTOM.

3DTOM is limited to about 8000 raypaths, but GeoTomCG is limited only by the computer memory. GeoTomCG is written in Microsoft Visual C++, and so provides a more graphical user interface than 3DTOM. GeoTomCG offers many advantages:

1. Faster inversions.
2. Number of raypaths and grid points limited only by computer memory.
3. Curved-ray attenuation tomography using raypaths from velocity tomography.
4. Graphical editing of initial velocity model.
5. Graphical display of residuals and other parameters as a function of raypath identifier.
6. Reflected rays, if reflection plane is known.

## INSTALLING AND STARTING GEOTOMCG

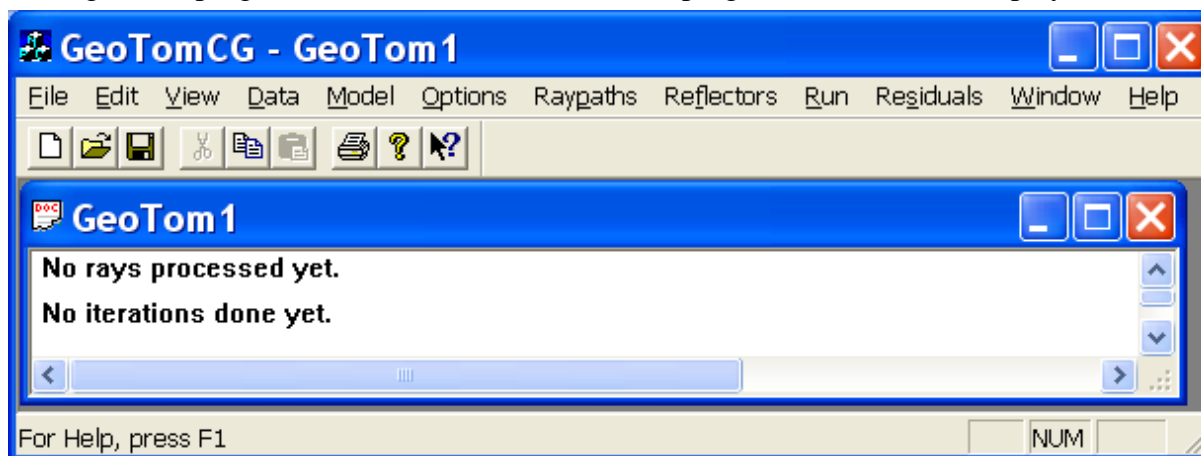
The GeoTomCG CD contains a Microsoft Installer (msi) program for Win2000 and later. Run SetupGeotomCG.exe in the subdirectory GeoTomCG to start the installation. Choose a name for the subdirectory in which GeoTomCG should be installed. The installation program will install the GeoTomCG program, sample files, and user's instructions in the chosen subdirectory, and will add GeoTomCG to the program list. If installation fails, contact the author. If the security settings on your computer prevent writing practice files in the default Program Files folder after installation, copy the example data to a new folder.

Sample files include:

Demo1.3dd and Demo1WithLabels.3dd, standard 3D data files;  
 Demo1WithLabels.3dw, like Demo1WithLabels.3dd plus a quality weighting factor;  
 Demo1.3dm and Demo1.3mq, initial velocity models (grid points) without (\*.3dm) and with (\*.3mq) quality factors, with uniform velocity for Demo1 data;  
 Data.2dd, 2D data file;  
 Reflex2D.TOM and Reflex3D.TOM, in 2D and 3D formats for traveltimes from ReflexW;  
 3DCylSynth\*.\*, data, initial uniform and calculated velocity models for 3D cylinder display;  
 \*.tam, several combined time and amplitude data files for practicing processing amplitudes (see chapter "Illustrated Examples of Amplitude Processing");  
 AmpExamp.xls, spreadsheet showing how to process amplitudes for GeoTomCG input;  
 Reflect.ref, file specifying a reflection plane;  
 \*.CLR, four color-spectrum files for displaying results.

If sent by e-mail, the files are in a password-protected compressed zip file, called GeoTomCG(version).zip. The password is "geotom". Some virus protection programs block compressed files containing an executable file unless they are protected by a password. Make a subdirectory for unzipping GeoTomCG(version).zip on your hard disk. After unzipping it, run SetupGeotomCG.exe to start the installation and follow the other instructions above.

Start GeoTomCG in the usual manner for Windows programs, by either clicking on the name in the Windows icon or program list or switching to the GeoTomCG subdirectory and double-clicking on the program name GeoTomCG.exe. The program will start and display this menu.



## GENERATING AND EDITING INPUT FILES

The measured quantity is called time in these instructions, but it could be the processed amplitude. Similarly, the calculated parameter is called velocity, but the attenuation coefficient can be calculated if amplitudes are measured. Instructions for processing amplitudes for attenuation tomography are in the chapter “Special Procedures for Attenuation Tomography.”

### Data Files (Source and Receiver Coordinates, Traveltimes)

The user must provide an input data file in one of several formats described below: a 3D file format with one measured variable (usually time), a combined 3D format containing both time and amplitude, or a 2D format. The companion time-picking program TomTime can prepare a data file in either the standard 3D (\*.3dd) or the combined (\*.tam) formats described below. Coordinates are double-precision floating point numbers, so 10-digit GPS coordinates in the form 1234567.123 can be used. GPS coordinates in other forms must be converted to that form to be read and displayed correctly.

#### Synthetic Data File

A synthetic data file in the standard 3D format described below can be prepared with the **Generate data file** under the **Data** menu. If TomTime is not available, generating a synthetic data file can reduce the typing of coordinates for a real data file. After the user clicks **Data**, **Generate data file**, the dialog box shown to the right will be displayed. The edit boxes in rows labeled **Minimum** and **Maximum** are for entering the minimum and maximum allowed X, Y, and Z for all sources and receivers. The edit boxes below them are for entering the coordinates of the first and last source position and the X, Y, and Z increments of source coordinates. The increment is the change from one position to the next. The edit boxes near the bottom are for the receiver X, Y, and Z coordinates or offsets from the source. Click on **Coor** or **Offset** to select which variable to enter. **Coor** is often simpler to use and is the default. For the most

Generate data file, assume constant velocity

Borehole Direction  
 X  Y  Z (vertical)  Other

Panel #  Source hole label  Receiver hole label

Upper and lower limits for source and receiver positions

	X	Y	Z
Minimum	<input type="text" value="0"/>	<input type="text" value="0"/>	<input type="text" value="5"/>
Maximum	<input type="text" value="20"/>	<input type="text" value="0"/>	<input type="text" value="28"/>

Velocity  
 Velocity for calculating times

Source coordinates

	X	Y	Z
First	<input type="text" value="0"/>	<input type="text" value="0"/>	<input type="text" value="5"/>
Last	<input type="text" value="0"/>	<input type="text" value="0"/>	<input type="text" value="28"/>
Increment	<input type="text" value="0"/>	<input type="text" value="0"/>	<input type="text" value="0.5"/>

Number of receivers for each source

Add another panel

Receiver coordinates or offsets from each source

	X	Y	Z
First rec, each src	<input type="text" value="20"/>	<input type="text" value="0"/>	<input type="text" value="5"/>
Offset increments	<input type="text" value="0"/>	<input type="text" value="0"/>	<input type="text" value="1"/>

OK  
Cancel



commonly generated data file consisting of vertical boreholes with constant X and Y, the default **Coor X** and **Y** will be most convenient. Default values for borehole information are shown above. They will generate a synthetic data file with a source borehole at X=0, Y= 0 and 24 receivers in a borehole at X=20, Y= 0. The source Z range is 5 to 28 and the source Z spacing is 0.5. The receiver spacing is 1, and all 24 receivers receive signals from each source position. The assumed velocity for calculating traveltimes can be entered, and is 3 for these data.

Offsets are useful when the receiver positions move with the source, as for testing piles. Offsets are relative to the source, not the actual coordinates of the receivers. You can enter the number of receiver positions for each source position and the receiver increment, which is the change in receiver coordinate from one position to the next. Increments can be positive or negative. The parameters shown to the right will generate a data file with sources at X=0, Y=0.6, and Z ranging from 2 to 48 with Z spacing 1. The receivers are at X=1.2, Y=0. The five receivers move with the source so one receiver is at the source Z, with two above and two below at Z spacing 1.

Generate data file, assume constant velocity

Borehole Direction  
 X  Y  Z (vertical)  Other

Panel #  Source hole label  Receiver hole label

Upper and lower limits for source and receiver positions

	X	Y	Z
Minimum	<input type="text" value="0"/>	<input type="text" value="0"/>	<input type="text" value="0"/>
Maximum	<input type="text" value="1.2"/>	<input type="text" value="0.6"/>	<input type="text" value="50"/>

Source coordinates

	X	Y	Z
First	<input type="text" value="0"/>	<input type="text" value="0.6"/>	<input type="text" value="2"/>
Last	<input type="text" value="0"/>	<input type="text" value="0.6"/>	<input type="text" value="48"/>
Increment	<input type="text" value="0"/>	<input type="text" value="0"/>	<input type="text" value="1"/>

Number of receivers for each source

Velocity  
 Velocity for calculating times

Receiver coordinates or offsets from each source

	X	Y	Z
First rec, each src	<input type="text" value="1.2"/>	<input type="text" value="0"/>	<input type="text" value="-2"/>
Offset increments	<input type="text" value="0"/>	<input type="text" value="0"/>	<input type="text" value="1"/>

Add another panel  
 OK  
 Cancel

You can enter source and receiver borehole labels of up to three characters in the corresponding edit boxes near the top right. A label will be displayed at one end of the corresponding borehole in tomograms. You may wish to adjust the label position or add a label for the other end of the borehole. The following section, Data in Standard 3D Format, One Measured Variable, provides additional explanation and examples of borehole labels.

To generate a data file with only one panel (source-borehole to receiver-borehole pair) click **OK** and provide a file name when requested. To include more than one panel, enter the coordinates and other information for the first panel, then click **Add another panel** and provide a file name when requested. The dialog box will reappear so you can enter coordinates and other information for the additional panels. When you have entered the information for the last panel, click **OK**. The panel number will increase by 1 each time you add a panel to help remind you for which panel you are entering data. The panel number and the borehole direction are not used by GeoTomCG in calculations.

### Data in Standard 3D Format, One Measured Variable

The standard format can represent either 3D or 2D data. For 2D data, the source and receiver Y coordinates should be included as any constant number. The file extension is 3dd.or 3dw. The 3dw files are like the 3dd files plus a weighting factor that can be used to diminish the weighting of poorer data. The corresponding example files are Demo1.3dd and Demo1.3dw, respectively. The file format is:

Two lines of general information, up to 80 characters in each line.

One line for each raypath, containing

Ray identifier Source X, Y, Z Receiver X, Y, Z Time (or amplitude) Weighting (in 3dw)

The ray identifier for direct rays is a positive integer. The ray identifier for any reflected rays must be a negative integer. They do not need to be sequential, but making them unique is advantageous when plotting ray identifier versus residual (see “Plotting Variables in Residuals Files”). The source and receiver X, Y, and Z coordinates, the time (or amplitude), and the weighting factor are floating point numbers. These numbers do not need to be in particular columns, but must be separated by spaces, commas, or tabs. The weighting factor multiplies the correction factor when fitting data, so it should be 1 for good data and less than 1 for poorer data.

You can designate optional labels of up to three characters for boreholes and specify where the labels should be displayed. The labels can be entered on any of the lines of the data file, but putting them immediately after the two header lines is usually most convenient. The example below shows a data file with four labels, 1, 2, 3, and 4, to be displayed at the top and bottom of the boreholes, followed by the first two lines of the raypaths.

Output file from CSL software from Olson Instruments, Inc. FAT is first-arrival time.

Ray#	SourceX	SourceY	SourceZ	ReceiverX	ReceiverY	ReceiverZ	FAT (ms)
-32000 1	0.243	2.319	0.0				
-32000 2	1.239	1.236	0.0				
-32000 3	0.0	0.0	0.0				
-32000 4	-0.841	1.066	0.0				
-32000 1	0.243	2.319	-24.5				
-32000 2	1.239	1.236	-24.5				
-32000 3	0.0	0.0	-24.5				
-32000 4	-0.841	1.066	-24.5				
1	0.243	2.319	-17.703	1.239	1.236	-24.493	0.484
2	0.243	2.319	-17.892	1.239	1.236	-24.493	0.476

If -32000 is entered as a Raypath Identifier, GeoTomCG will read the next character as the label and the following three numbers as the X, Y, and Z coordinates where the label should be displayed. The labels should follow one blank space after the -32000. If labels overlap on the display, the Z coordinates can be incremented. The user can specify as many labels as desired. The labels can be up to three characters long, but single characters reduce overlapping. Borehole labels are optional. No labels will be displayed if no lines start with -32000.

### Data in Combination 3D Format, Two measured Variables

The combined file format containing both time and processed amplitude data is convenient when both time and amplitude have been measured. The file extension is tam (time, amplitude).

The only change in the format is that each raypath line is:

Ray identifier      Source X, Y, Z      Receiver X, Y, Z      Time      Processed amplitude

Amplitude data such as the amplitudes from the time-picking program TomTime must be processed to be analogous to time (see “Special Procedures for Attenuation Tomography”) before they can be used as input for GeoTomCG.

### Data in 2D and 3D TOM Format

GeoTomCG can read data in the 2D and 3D TOM formats generated by picking traveltimes with Reflex. The format for 3D is:

Time      Code      Source X, Y, Z      Receiver X, Y, Z

The format for 2D omits the Y coordinates, so GeoTomCG assigns 0 to them. Code is not used by GeoTomCG. Reflex uses the file extension TOM for both 2D and 3D, so the two example files both have the extension TOM. The user must choose the file to match the selected option.

### Data in 2D Alternative Format

Several customers with 2D data in an alternative format requested this option. The file extension is 2dd. This format is:

The number of source positions.

First source X and Z      Number of receivers (Nrec) associated with that source

Nrec lines containing

Receiver X and Z      Time      Weighting.

The data for the remaining sources follow in a similar manner.

GeoTomCG assigns 0 to the source and receiver Y coordinates.

The data weightings multiply the correction factors when fitting data, as for 3dw files.

### Plotting Variables in Data Files

The variables in data files can be plotted to help detect problems. First open a data file by clicking **Data**, then click the desired type of data file. A standard Windows file selection dialog box will appear. Select a data file and click **Open**. The first 1000 lines of the file are shown so you can see if it is the correct file and was read correctly. The program displays only 1000 lines because it is often used with large data files that take a long time to display, and any problems in the data file are usually evident in the first 1000 lines.

Next click **Data, Plot**, and the dialog box shown to the right with plotting options will appear in the upper left corner of the screen.

The user can select variables for the horizontal and vertical axes of the plot, and can enter new values for their ranges. The plot axes span the ranges in the dialog box, and may be somewhat larger to make the tic marks occur at intervals of 1, 2, or 5. The separate entry for ray identifier range allows the user to select a combination of a ray identifier range and some other variable for the horizontal axis. When the variables and ranges are satisfactory, click **OK** to display the plot. The Demo1.3dd data plot from the settings in this dialog box is shown below.

Plot data

Horizontal axis variable

Source-rec distance

Source  X  Y  Z

Common-receiver gather?

Receiver  X  Y  Z

Common-source gather?

Angle from horiz. deg

Ray identifier

Horizontal axis range

Minimum 6.31392

Maximum 17.9977

Vertical axis variable

Measured time or amp

Str-line vel or 1/atten

Vertical axis range

Minimum 0.3525

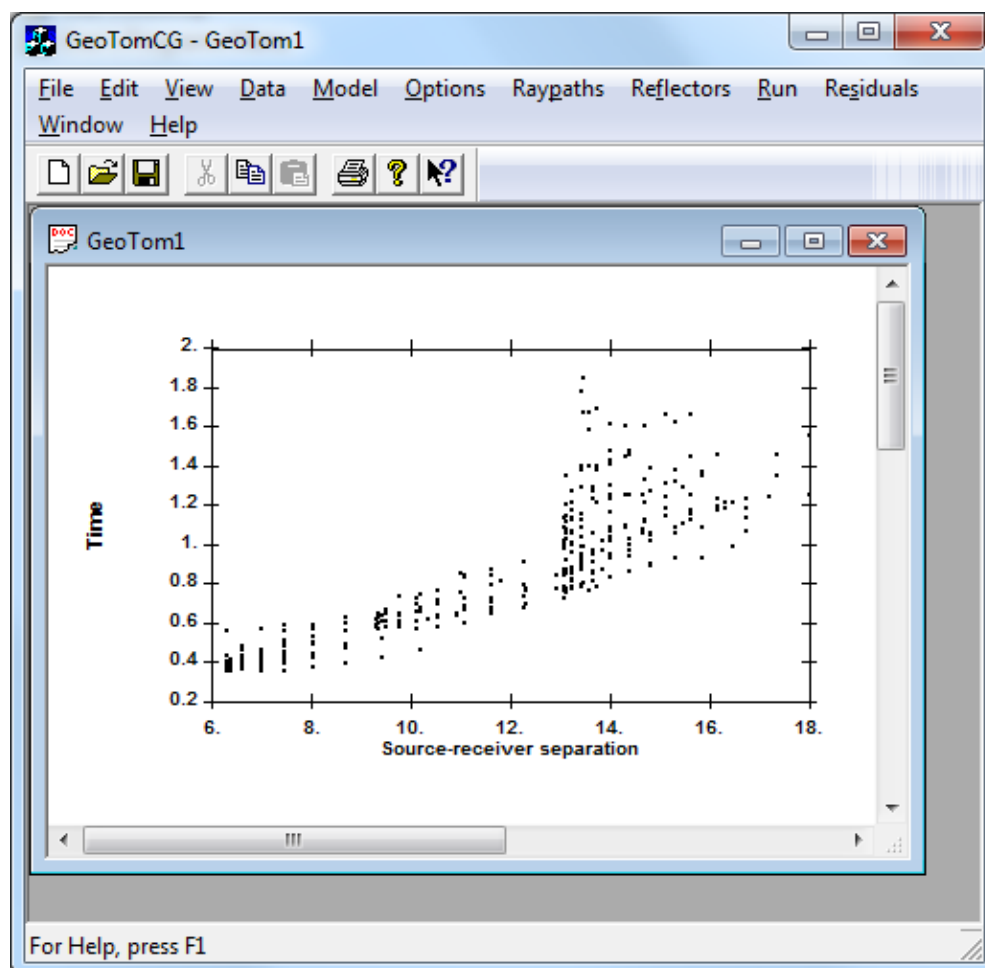
Maximum 1.844

Ray identifier range

Minimum 0

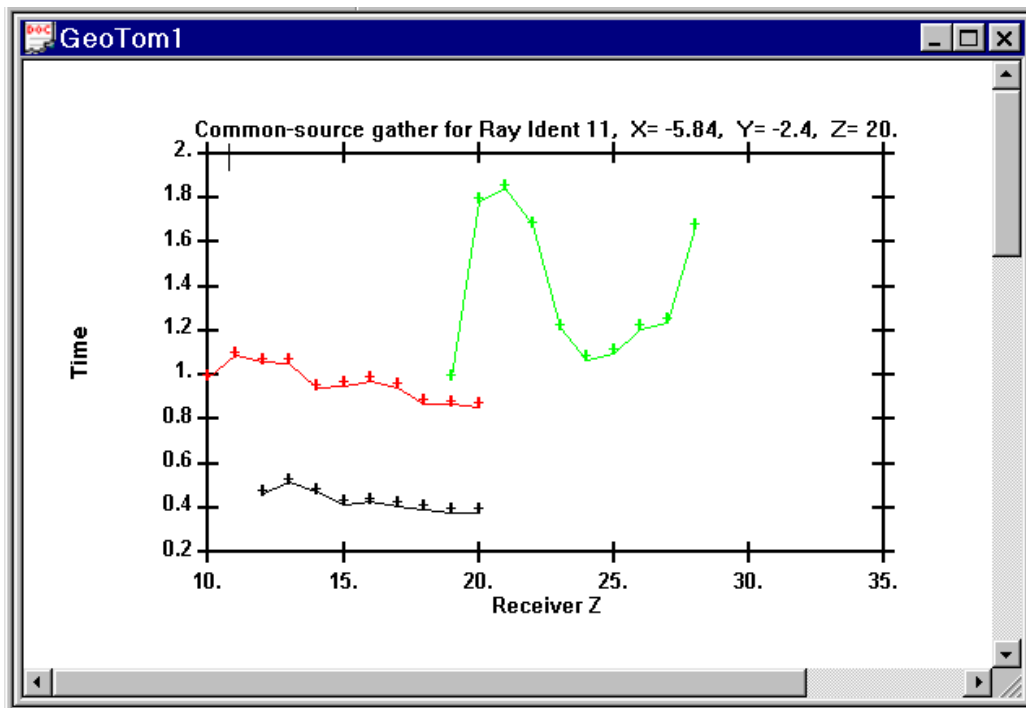
Maximum 30000

OK Cancel



When the variable on the horizontal axis is the ray identifier, the user can determine the ray identifier for a particular point by right-clicking on the plot to activate this option, then left-clicking on or just to the right of the point and, if necessary, using the left and right arrow keys to move the indicator by one point. The small line at the top shows the approximate position and the caption at the top shows the exact identifier. These are ray identifiers, not the ray subscripts in the array, so the user can assign meaningful ray identifiers and select different sections of the data for plotting.

The user can display common-source or common-receiver gathers by selecting the appropriate variable for the horizontal and vertical axis and clicking on the corresponding check box for the gather. For example, to display common-source gathers with vertical boreholes, you would usually select Receiver **Z** for the horizontal axis and time for the vertical axis, then click on the **Common-source gather** check box. The corresponding plot is shown below for the Demo1.3dd sample data set.



In 3D data sets, each source position can have raypaths going to several different sets of receivers, so the different planes for the source are represented with different colors. Black points and line are to the first set, red are to the second set, and green are to the third set. A fourth and later planes would be represented by blue. The ray can be selected in the same manner as raypaths when displaying the **Ray Ident** on the horizontal axis. The thin line on the top axis (near the left end of the top axis in the plot above) represents the raypath number, even though the horizontal axis is receiver **Z**. The user can change the raypath and hence the source by right clicking to set the plot to respond to the mouse, then left-clicking at the approximate position of the raypath. Then use the left and right arrow keys to move the raypath one at a time. For example, if the source you want is in a raypath about in the middle of the data set, then right-click followed by a left click near the middle of the top axis, then use the left and right arrow

keys to select the exact source. You can also use repeated left mouse clicks, but getting exactly the source you want with the mouse can be difficult in large data sets.

Plotting ray angle versus velocity can help to reveal anisotropy or layering. If layering is present, the maximum velocity is usually in line with the layering. If you select **Data, Plot** after reading a data file and then select **Angle from horiz** for the horizontal axis and **Str-line vel** for the vertical axis, GeoTomCG will open both a 2D plot and a 3D plot. In the 2D plot, the ray angle is in degrees from a horizontal (XY) plane for a straight ray traveling directly from source to receiver. A positive angle means the source-to-receiver path is upwards. If the default setting of **Down** is left under **Options, Pos Z axis up or down**, then upward corresponds to decreasing Z (e.g. depth). The 3D plot is in a separate window. It will cover the 2D plot initially, but can be dragged to the side to view both plots. The 3D plot will continue until you close it by clicking the X in the upper right or pressing the Escape key.

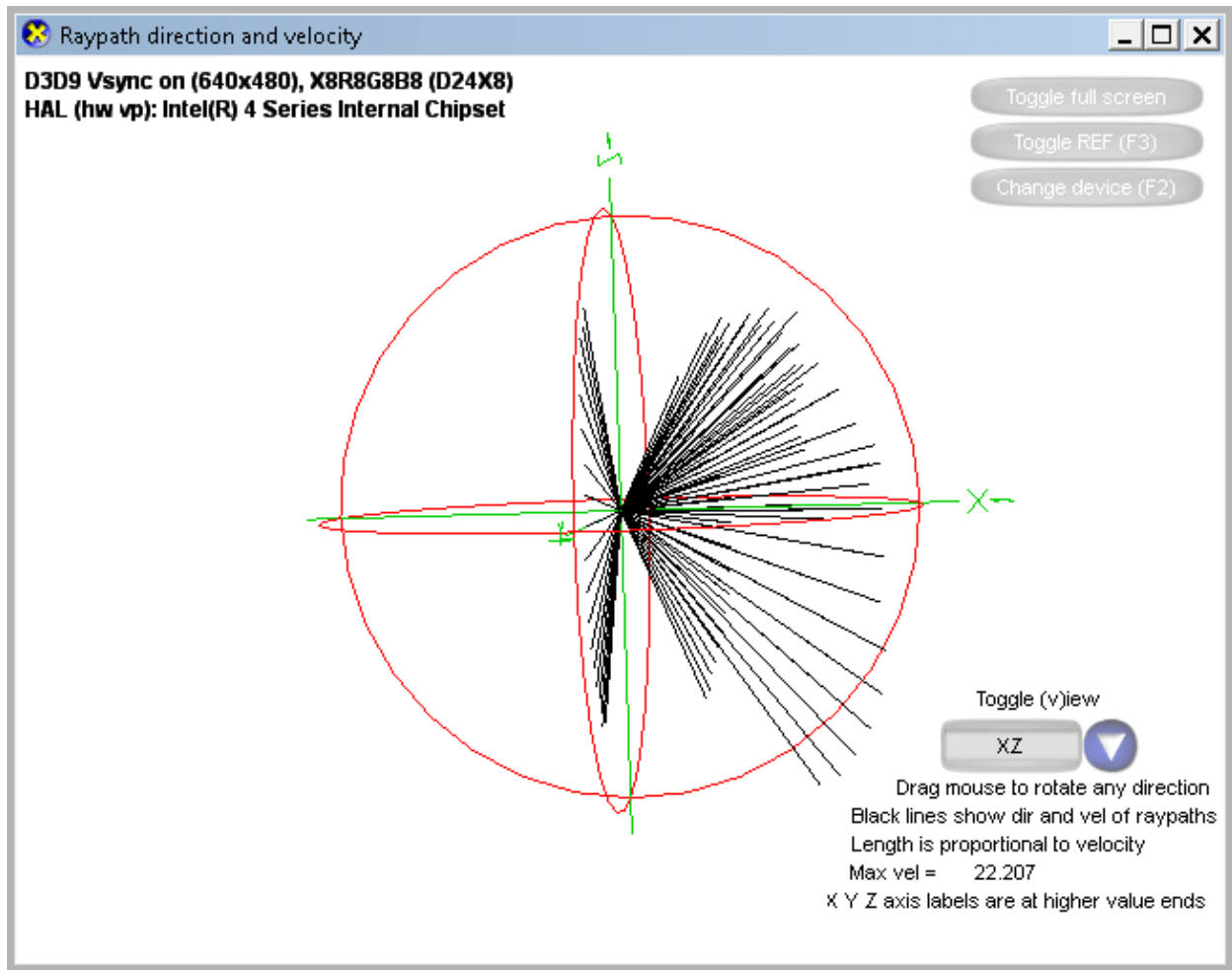
The 3D plot is generated by DataDisplayDir3D.exe created using Microsoft Direct3D. The installation program should automatically copy the following files to your Windows folder: DataDisplayDir3D.exe, DataDisplayDir3D.fx, d3dx9\_42.dll, d3dx10\_42.dll, and D3DCompiler\_42.dll. If you get an error message that says “Windows cannot find DataDisplayDir3D.exe,” then copy those five files from the Program Files, GeoTom, GeoTomCG\_Jan2015 folder if you used the default installation (or wherever you installed it) and paste them into the Windows folder. Do not make a separate folder for them. You will need to do that at most only once. DataDisplayDir3D.exe is called by GeoTomCG after GeoTomCG creates a very small subdirectory DataDisplayDir3D on your C drive and writes a two-line configuration file DataDisplayDir3D.ini that DirectDisplayDir3D.exe reads. The configuration file contains the name of the data file.

The purpose of the 3D display is to provide a fully 3D rotatable view of raypath directions vs. velocity to check for anisotropy. The 2D plot of raypath angle from horizontal vs. velocity is often adequate to reveal anisotropy because anisotropy is often related to horizontal or nearly horizontal layering with the direction of maximum velocity parallel to those layers. In some special situations, however, a 3D view can be helpful. For example, if the direction of highest velocity is tilted at 45 deg to horizontal and there is a source borehole between two receiver boreholes, then the direction of both the highest and lowest velocities can appear at both pos and neg 45 deg from horizontal. In this case, the 2D plot shows a spread of velocities, but does not indicate anisotropy. You may also find the 3D display helps to visualize the angle of anisotropy.

Both the 2D and the 3D displays use straight raypaths from source positions to receiver positions for calculating the raypath angles and the velocities. This straight-line velocity = source to receiver distance/measured time, and so represents the average velocity over each raypath without considering curvature. These displays can be viewed before the tomographic inversion, so the lengths of curved raypaths are not available.

The 3D display can be rotated in any direction by dragging the mouse. The raypath directions in the display are shown by the directions of the black lines relative to the blue X, Y, Z axes. These lines do not show the positions of sources or receivers. The length of each black line is proportional to the straight-line velocity for raypaths in the plane of the view. However, the display uses a perspective projection, so an object appears larger closer to the viewer. The red

rings in the 3 planes show the approximate maximum velocity. The green X, Y, and Z axes have labels on the positive end of each axis. The axis labels rotate with the display, so there are two labels for each axis, one in each of two planes. A label viewed on edge may look like a minus sign, but labels are on the positive end of each axis and there are no minus signs.



The 3D display of raypath angle vs. velocity for Demo1.3dd is shown above. The view plane is close to the XZ plane, but rotated slightly to the left and down to show all three rings. Pressing the v key toggles the view plane between the XZ, XY, and YZ planes. The directions of the black lines relative to the axes show the raypath directions. The length of the lines in the view plane is proportional to the velocity. The red rings are at approximately the maximum velocity for rays in the view plane. This data display is a recent feature, and suggestions for improvements sent to sales@geotom.net are welcome.

### Removing Outliers in Data Files

This feature allows the user to generate a new data file with specified raypaths from the original removed, using several criteria to remove outliers or other suspect raypaths.

After opening a data file, click **Data, Remove outliers**. A dialog box will appear with check boxes for selecting which parameters are used as selection criteria and text boxes for entering the acceptable ranges. The available parameters are ray identifier, straight-line velocity (or 1/atten coefficient), time (or amplitude), and ray angle from horizontal in degrees.

To include a parameter in the selection, select the corresponding check box. To select a corresponding range, enter the values in the edit boxes. Note that the range for **ray ident** has the opposite effect of the ranges for the other parameters. The rays with ray identifiers within the range will be removed. For all the other parameters, the rays will be retained if the corresponding value of the parameter is within the range. This was done because removing outliers based on ray identifiers usually involves removing several ray paths that are not at the extremes of the ray path identifier numbers. For velocity and the other parameters, the user usually wants to remove rays with unusually low or high values. If a parameter is not selected as a criterion, then its range does not need to be specified.

After choosing the selection parameters and the ranges for the selected parameters, click **OK**. A standard **Save As** dialog box will appear to allow the user to select a name for the new data file. Enter a name, then click **Save**. A new data file will be generated in the standard 3dd format. The new data file is not opened automatically. It can be opened by clicking **Data, Open data file (\*.3dd)**, and entering the name of the new data file.

Outliers can also be removed using residuals or normalized residuals after inversion, as explained in the later chapter "Plots and Files from Residuals Files," section "File of Measured Times with Outliers Removed."

### Rotating Coordinates

The X and Y coordinates in data files can be rotated about the Z axis. This option can be convenient to make two boreholes and the raypaths between them all be displayed in the same plane. After opening a data file, click **Rotate Coordinates** in the **Data** menu. A dialog box will appear for selecting the desired angle and X Y position of the center of rotation. Enter the angle in degrees. A positive angle makes a counterclockwise rotation. The default center of rotation is the midpoint of the range of X and Y for sources and receivers, but you can select different X and Y. Click **OK**, a then enter a name for the new data file with rotated coordinates. The rotated data file will be in the 3dd format if the original file did not use data weighting and 3dw if it did.



## Initial Velocity Model

A file containing the initial velocity estimate for each point of the model grid must also be opened before running the inversion. However, the program helps the user prepare it as described below, so the user does not need to prepare this file before starting the program.

### Generating the Model

A model file must be opened or GeoTomCG will not perform an inversion. A default model is created by GeoTomCG, but it may not be ideal. You may want to revise it to obtain, for example, integer heights and widths for voxels.

First open a data file in any of the data formats. The data file is used to generate a grid of points that covers the source and receiver coordinates. Click **Data**, then **Open data file** or **Combined, Read Time** or **Combined, Read Amplitude** or **Import 2D data**. A standard Windows file selection dialog box will appear. Select the data file you want and click **Open**. The first 1000 lines of the file will be shown to allow you to check if it is the correct file and was read correctly.

Next click **Model, Generate model**. The dialog box shown below with information about the current model (the default model if no other model has been opened) and boxes for entering new model parameters appears.

Source and receiver positions					
	Smallest	Largest			
X	-5.84	7.5			
Y	-3.8	9.29			
Z	10	33			
Grid XYZ ranges and numbers of grid points					
	Smallest	Largest	Number		
X	-7	9	9	X	0
Y	-5	11	9	Y	0
Z	8	34	14	Z	0

Uniform velocity: 11.0224

Uniform anisotropy: 1

Anisotropy dir: X: 0, Y: 0, Z: 0

Buttons: OK, Cancel

The smallest and largest X, Y, and Z coordinates for all sources and receivers are shown, to help choose appropriate limits for the grid points. This information is given in non-white boxes, indicating that it cannot be edited. The smallest and largest X, Y, and Z coordinates for the current grid-points and the number of points in each direction are shown in white boxes, indicating that these parameters can be edited. Click on any entry that you want to change and enter a new value. The default grid extends beyond the source and receiver positions to allow room for curved rays. The velocity in the model generated in this procedure is uniform. The

desired velocity can be entered in the box labeled **Uniform velocity**. The default velocity is between the average and lowest straight-line velocity calculated from the data file.

### Specifying Anisotropy

This section briefly describes how to specify anisotropy. The later chapter titled "Checking For and Applying Anisotropy" provides more details on how to use the results of preliminary inversions. You would probably start with no anisotropy (magnitude = 1.0) for the initial inversions.

The **Generate Model** dialog box provides boxes for entering anisotropy values. GeoTomCG does not invert for anisotropy, but allows the user to test whether anisotropy improves the fit. The anisotropy is associated with the grid points, so the model grid-point file contains the anisotropy information. A line containing @AnisotDir tells the program that the next line will give the X, Y, and Z components of the anisotropy direction vector. The components do not have to be normalized. The anisotropy direction is perpendicular to the plane of maximum velocity, and is uniform for the entire grid. The anisotropy direction can also be specified by editing the model file with a word processor.

If anisotropy will not be used, leave the default 0 for all of the components of the anisotropy direction vector, so the anisotropy direction components are:

0.0 0.0 0.0

Then anisotropy calculations are not performed. Anisotropy calculations slow execution, so setting the anisotropy direction to 0 0 0 improves efficiency if anisotropy will not be used.

If the anisotropy direction is vertical (horizontal layering) the X, Y, Z components are:  
 $\sin(0^\circ)$   $\sin(0^\circ)$   $\cos(0^\circ)$  or 0.0 0.0 1.0.

If the plane of maximum velocity has a dip of  $30^\circ$  with a strike along the Y-axis such that Z decreases from source to receiver, then the X, Y, Z components are:

$\sin(30^\circ)$   $\sin(0^\circ)$   $\cos(30^\circ)$  or 0.50 0.0 0.866

The magnitude of the anisotropy is given by an additional parameter for each point in the model file, following the point constraint. The magnitude is the ratio (velocity perpendicular to the anisotropy direction/velocity parallel to the anisotropy direction). Thus, for the common case of horizontal layering with the vertical velocity smaller, the direction is 0 0 1 and the magnitudes are greater than 1. This is done because anisotropy ratios are often given as the ratio (velocity in plane of maximum velocity /velocity in direction of minimum velocity).

The magnitudes of anisotropy can be set to any uniform value by entering that value in the box labeled **Uniform anisotropy**. The anisotropy at individual points can be changed by editing the model file with the graphical model editor described below.

### Saving the Model

To save the generated model, click **OK**. You will be asked to enter a name for the file in a standard **Save As** dialog box. The model will be saved with the extension 3mq to indicate it contains the columns for Data quality for points and Slowness residual/slowness. The format for lines containing information about individual grid points is:

X Y Z Velocity Constraint Anisotropy Data quality Slowness residual/slowness

Data quality and Slowness residual/slowness are set to 1 for the initial model. They are calculated during the inversion. The initial values are not used in the inversion. Data quality for each grid point is calculated using the Data quality for raypaths. The generated model is not loaded automatically. To use the generated model, you must open it after you have saved it. For backward compatibility, GeoTomCG can read model files in the previous 3dm format without Data quality for points and Slowness residual/slowness, but all new model files are saved in the 3mq format.

### Editing the Model

You have the option of specifying values at grid-points for velocity, constraints, and anisotropy. GeoTomCG provides flexible options for constraints, described in the following section “Specifying Constraints and Regularization.” Users may find the combination of compact notation and flexibility confusing at first. To begin, users can leave all point constraints at the default value of 0.0.

The grid-point velocities, constraints, and anisotropies can be changed with a word processor, but a more convenient method is to use the graphical grid-point editor in conjunction with the **Model, Display model** option. It provides options for selecting which of the three grid-point variables will be set and for drawing lines on the displayed model to show in which areas they should be set to specified values.

First display a model. GeoTomCG displays colors in small rectangles representing the velocity (or reciprocal of attenuation coefficient) at the center of each rectangle. After opening a model, select **Model, Display model**. A dialog box will appear in the upper left corner of the screen. Select a 2D slice plane with the radio buttons. Click **Show slice**. The slice will be shown in a scrollable window. Additional information on displaying the model is given in “Displaying tomograms.”

Two dialog boxes can be used for editing. One is for applying changes in areas or volumes in the grid that are drawn by the user or for applying changes in horizontal layers. The other allows the user to apply changes between two planes that can be tilted in any 3D direction.

## Drawing on Grid or Applying Horizontal Layers

Select **Model**, **Edit model**. This dialog box will appear in the upper left of the screen. The upper three areas provide very versatile editing capabilities but require drawing with a mouse. The next two areas provide convenient methods for specifying vertical gradients and horizontal layering.

In the upper three areas, select the combination of point parameters you want to set to values that you will specify in the edit boxes. **None** means that no values of that kind will be set. **Slice** means that only the points in the slice selected for the model display within the bounding lines drawn by the user will be set. **Through** means that all points in the volume extending on both sides of the displayed slice within the bounding lines will be set. For example, if the user draws a rectangle on the displayed slice and selects **Through**, then values at points within the rectangular solid extending on both sides of the displayed slice will be set. Then enter a value in the edit box for each of the point parameters that you want to set.

The next step is to draw the bounding lines for the editing. Click on **Do more editing before saving**, then click **OK**, and the model slice will be shown ready for drawing lines. Click the left mouse button to make the first point and hold the left mouse button down as you move the cursor away from the first point. A line will appear for “rubber-band” drawing. Move the cursor to the desired point and release the left mouse button. Press the left mouse button again and move it with the button down to select the next point. Continue until the bounding lines surround the area that you want to change. The closure does not have to be exact. Then press and release the right mouse button. The program will close the bounding

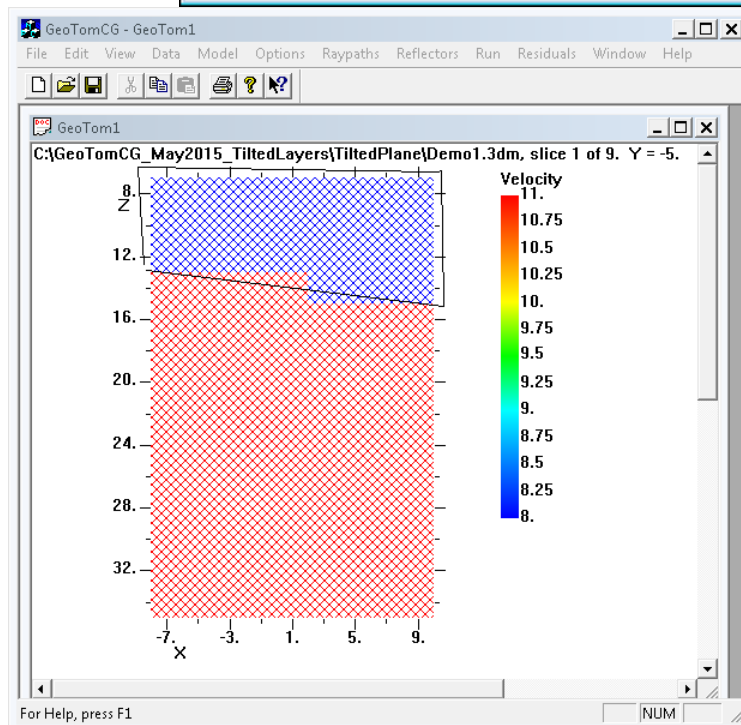


figure as if a line were drawn from your last point to the first point. The specified changes will then be applied to the points within the bounding figure. In this example, a velocity of 8 was added in a slanted top layer to the Demo1.3dm with original initial velocity uniform 11.

The user can apply a uniform vertical velocity gradient by entering two velocity values and the corresponding Z values in text boxes in the **Set uniform vertical velocity gradient** area of the **Edit model** dialog box, then clicking **Apply uniform vertical gradient** before clicking **OK**. The velocity gradient is applied over the entire horizontal extent of the grid

The user can apply horizontal layering in horizontal slices by entering the starting and ending Z values and the uncertainty factor in text boxes in the **Set horizontal layering in horizontal slices** area of the **Edit model** dialog box, then clicking **Apply horizontal layering** before clicking **OK**. Horizontal layering is applied as a set of point constraints over the entire horizontal extent of each horizontal slice within the specified Z range. These point constraints are described in the next section under Point by Point, type (c), and the uncertainty factor is described in the paragraph following (c). The positive integer assigned to each layer is the horizontal slice number. For example, if the starting Z for horizontal layering is the minimum grid Z, then the corresponding horizontal slice number is 1.

This editing procedure can be repeated as many times as you want. When you are satisfied with the edited model, click **Save edited model this time** and click **OK**. A dialog box will appear asking you to provide a name for the newly edited model file. Saving the edited model clears the bounding lines from the display.

### Applying 3D Tilted Layers

The newest set of editing options allows the user to apply gradients, layering, or anisotropy between parallel planes tilted in any direction except vertical. The user first specifies the direction of a plane by entering the coordinates for three points. This plane is called the direction plane in the explanation below. It extends across the entire grid. Then the user can edit a velocity model in any of three ways between two planes parallel to the specified plane. Those planes are called option planes in the explanation below. The Z position of each of the two option planes is specified by entering the Z coordinate for the X and Y at point 1. The three options are:

1. Apply a linear velocity gradient between the two planes.
2. Apply layering constraints between the two planes, similar to applying layering between two horizontal planes.
3. Specify a uniform strength of anisotropy between the two planes. The direction cosines for the perpendicular to the planes are calculated and written in the saved velocity model. Users may find that specifying the coordinates for a plane is easier to understand than the direction cosines of the perpendicular to the plane.

To apply an option, open a velocity model file, then click **Model, Edit model, use tilted planes in 3D**. A dialog box like the one to the right will appear. This example is for the example file Demo1.3dm.

When this dialog box is first opened, default values are provided. The direction plane default Point 1 has X and Y at their minimum values. The default Point 2 has X and Y at their maximum and minimum, respectively. The default Point 3 has X and Y at their minimum and maximum, respectively. The default Z values are the average of minimum and maximum. For the option planes, the default Z values are at intermediate values. The default velocities for layers 1 and 2 are intermediate velocities. If the velocity is uniform, the default layer velocities are the same, so the default values for this example are both 11.

The dialog box is titled "Make 3D tilted layers, options for velocity gradient, layering, and anisotropy". It contains the following sections:

- Enter three 3D points to define a plane**: A note states "Planes for layering, gradient, and anisotropy options below are parallel, with same point 1 X and Y." Below this are three rows of input fields:
 

Point 1 X Y Z	-7	-5	21
Point 2 X Y Z	9	-5	21
Point 3 X Y Z	-7	11	21
- Set linear velocity gradient between two 3D planes**: Two rows of input fields for "Plane 1 point 1 Z" and "Plane 2 point 1 Z", each followed by a "Velocity =" field.
 

Plane 1 point 1 Z	18.4	Velocity =	11
Plane 2 point 1 Z	23.6	Velocity =	11

 An "Apply linear velocity gradient" button is located below.
- Set layering between two 3D planes**: Two rows of input fields for "Plane 1 point 1 Z" and "Plane 2 point 1 Z", and an "Uncertainty factor" field.
 

Plane 1 point 1 Z	18.4	Uncertainty factor	0.5
Plane 2 point 1 Z	23.6	Uncertainty 0.0 applies layering full strength, 0.9 weakly.	

 An "Apply 3D layering" button is located below.
- Set strength of polar anisotropy between two 3D planes**: A note states "Max vel is parallel to those planes. Strength = vel parallel/perpendicular to planes." Below this are two rows of input fields for "Plane 1 point 1 Z" and "Plane 2 point 1 Z", and a "Strength" field.
 

Plane 1 point 1 Z	18.4	Strength	1
Plane 2 point 1 Z	23.6	Apply anisotropy	

At the bottom of the dialog box are "Save edited model" and "Cancel" buttons.

As an example of editing Demo1.3dm, suppose cores show the following subsurface conditions. (The units for Demo1.3dm are ft and ms.)

a) Uniformly soft material with estimated seismic velocity of 6 ft/ms down to:

12 ft at X = -5 ft, Y = -3 ft,

16 ft at X = 7 ft, Y = -1 ft

18 ft at X = -3 ft, Y = 7 ft.

b) Increasingly competent rock to a depth of 6 ft below the bottom of the soft material.

c) Uniformly harder rock with an estimated seismic velocity of 14 ft/ms at depths greater than 6 ft below the bottom of the uniformly soft material.

Setting the coordinates of the direction plane to the core hole coordinates and layer depths is convenient. For this example, set the coordinates for the direction plane as follows:

Point 1 X = -5, Y = -3, Z = 12.

Point 2 X = 7, Y = -1, Z = 16.

Point 3 X = -3, Y = 7, Z = 18.

Next, enter values for the option planes under **Set linear velocity gradient between two 3D planes**. For this example, start at the surface and work down, so set the velocity to 6 ft/ms everywhere in the grid above the direction plane. The linear gradient option applies a uniform velocity between two option planes if the velocity is equal for both planes. Option plane 1 is at the surface. Option plane 2 is coincident with the direction plane.

Set option plane 1, point 1 Z = -10, and velocity = 6.

Set option plane 2, point 1 Z = 12, and velocity = 6.

The exact value of plane 1 point 1 Z does not matter. It just needs to be shallow enough so all of the grid points above option plane 2 are included. GeoTomCG ignores points outside of the grid. Then the dialog box will be as shown below. Click **Apply linear velocity gradient** to apply this set of changes.

Make 3D tilted layers, options for velocity gradient, layering, and anisotropy ✕

Enter three 3D points to define a plane  
Planes for layering, gradient, and anisotropy options below are parallel, with same point 1 X and Y.

Point 1 X Y Z	-5	-3	12
Point 2 X Y Z	7	-1	16
Point 3 X Y Z	-3	7	18

Set linear velocity gradient between two 3D planes

Plane 1 point 1 Z	-10	Velocity =	6
Plane 2 point 1 Z	12	Velocity =	6

Set layering between two 3D planes

Plane 1 point 1 Z	18.4	Uncertainty factor	0.5
Plane 2 point 1 Z	23.6	Uncertainty 0.0 applies layering full strength, 0.9 weakly.	

Set strength of polar anisotropy between two 3D planes  
Max vel is parallel to those planes. Strength = vel parallel/perpendicular to planes.

Plane 1 point 1 Z	18.4	Strength	1
Plane 2 point 1 Z	23.6	<input type="button" value="Apply anisotropy"/>	

To apply a linear velocity gradient between the soft, 6 ft/ms material and the harder 14 ft/ms rock, enter the lowest and highest velocities and enter depths to make option plane 2 be 6 ft below option plane 1.

Plane 1 point 1 Z = 12 and velocity = 6,

Plane 2 point 1 Z = 18 and velocity = 14.

Then the dialog box will be as shown below. Click **Apply linear velocity gradient** to apply the gradient between the two tilted planes.

Make 3D tilted layers, options for velocity gradient, layering, and anisotropy ✕

Enter three 3D points to define a plane  
Planes for layering, gradient, and anisotropy options below are parallel, with same point 1 X and Y.

Point 1 X Y Z	-5	-3	12
Point 2 X Y Z	7	-1	16
Point 3 X Y Z	-3	7	18

Set linear velocity gradient between two 3D planes

Plane 1 point 1 Z	12	Velocity =	6
Plane 2 point 1 Z	18	Velocity =	14

Set layering between two 3D planes

Plane 1 point 1 Z	18.4	Uncertainty factor	0.5
Plane 2 point 1 Z	23.6	Uncertainty 0.0 applies layering full strength, 0.9 weakly.	

Set strength of polar anisotropy between two 3D planes  
Max vel is parallel to those planes. Strength = vel parallel/perpendicular to planes.

Plane 1 point 1 Z	18.4	Strength	1
Plane 2 point 1 Z	23.6	<input type="button" value="Apply anisotropy"/>	



The remaining step is to set the velocity to 14 ft/ms everywhere in the grid below the option plane 2 specified above. Apply the deep 14 ft/ms velocity by the following entries.

Plane 1 point 1 Z = 18, velocity = 14.

Plane 2 point 1 Z = 50, velocity = 14.

The exact value of option plane 2 point 1 Z does not matter. It just needs to be deep enough so all of the grid points below option plane 1 are included. GeoTomCG ignores points outside of the grid. Then the dialog box will be as shown below. Click **Apply linear velocity gradient** to apply this set of changes.

Make 3D tilted layers, options for velocity gradient, layering, and anistropy

Enter three 3D points to define a plane  
Planes for layering, gradient, and anistropy options below are parallel, with same point 1 X and Y.

Point 1 X Y Z	-5	-3	12
Point 2 X Y Z	7	-1	16
Point 3 X Y Z	-3	7	18

Set linear velocity gradient between two 3D planes

Plane 1 point 1 Z	18	Velocity =	14
Plane 2 point 1 Z	50	Velocity =	14

Apply linear velocity gradient

Set layering between two 3D planes

Plane 1 point 1 Z	18.4	Uncertainty factor	0.5
Plane 2 point 1 Z	23.6	Uncertainty 0.0 applies layering full strength, 0.9 weakly.	

Apply 3D layering

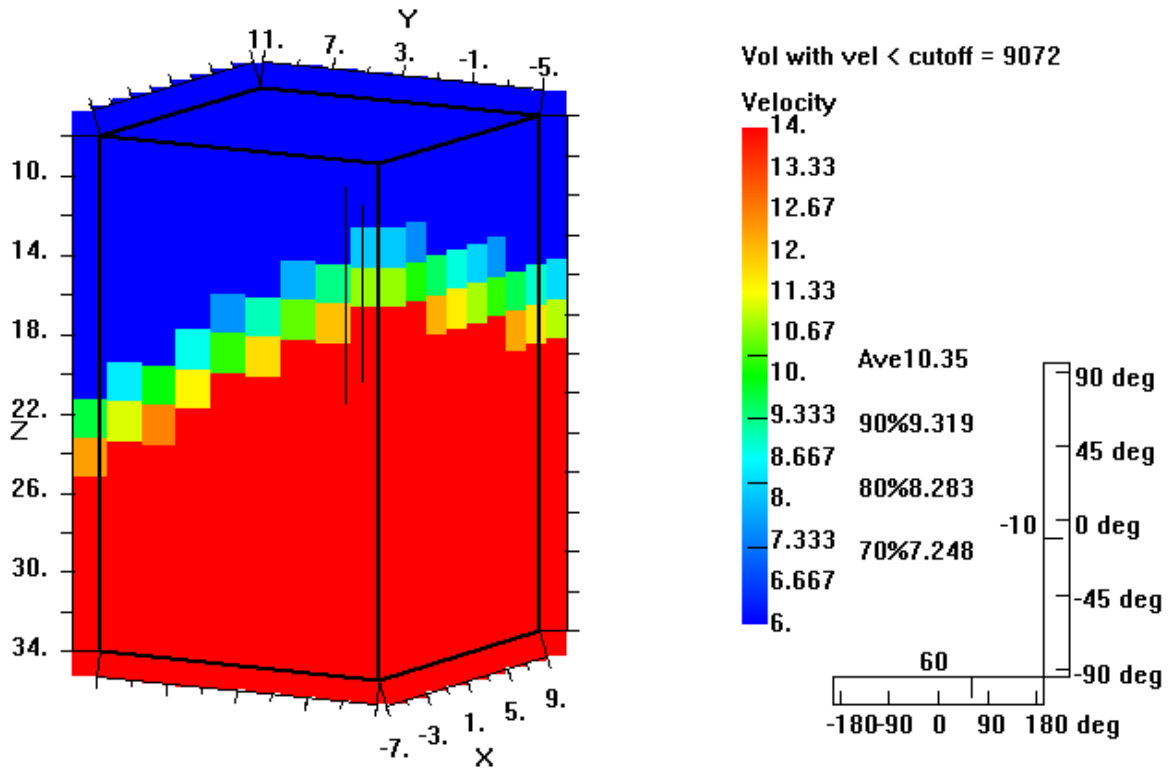
Set strength of polar anistropy between two 3D planes  
Max vel is parallel to those planes. Strength = vel parallel/perpendicular to planes.

Plane 1 point 1 Z	18.4	Strength	1
Plane 2 point 1 Z	23.6	Apply anistropy	

Save edited model    Cancel

Click **Save edited model** when you have finished editing with these options. A dialog box for saving a 3mq velocity model will appear. Enter a file name and click **OK** to save the edited initial velocity model.

The resulting velocity model file is shown in the 3D rectangle view below, with point 1 as the front corner. For this display, the Rays per point to fill color was set to 0, to make all surfaces solid color.



## Specifying Constraints and Regularization

Due to imperfect angular coverage in common source-receiver geometries, traveltimes data may not contain sufficient information to obtain a mathematically unique reconstruction of velocities (Menke, 1984; Jackson and Tweeton, 1994). It is important in such cases to use all other available information about the site. Application of constraints enables you to eliminate velocity solutions that are mathematically acceptable (in terms of satisfying the traveltimes data) but geologically unreasonable. For example, a nonunique inversion may result in a model with unrealistically high or low velocities or with a velocity structure that is unreasonable based on site knowledge.

In GeoTomCG, a distinction is made between constraints that are applied locally to portions of the grid, point by point (or zone by zone) and those that are applied over the entire grid. The former constraints are specified as part of the velocity model file. The latter are specified as part of setting the runtime inversion parameters in dialog boxes.

### Point by Point

Point constraints are useful for forcing the solution to match known velocities and for maintaining uniformity over groups of points (e.g., continuous layers). Because the latter type of constraint is often quite uncertain, point constraints are assigned an estimate of uncertainty, which is used as a weighting factor so that better-determined constraints have a stronger influence on the solution than do poorly determined ones. Such weighted constraints were referred to as “fuzzy” constraints in U.S. Bureau of Mines manuals for MIGRATOM (Jackson and Tweeton, 1994) and 3DTOM (Jackson and Tweeton, 1996). They are closely related to regularization. The regularization coefficient is a measure of the weighting given to the regularizing constraint, so the larger the regularization coefficient, the greater is the effect of the constraint. Conversely, the uncertainty is a measure of how much to reduce the weighting of the constraint, so the larger the uncertainty, the smaller should be the effect of the constraint. The uncertainty coefficient is used to allow more compact constraint nomenclature (see below), where a 0 after the decimal point indicates the constraint is applied full-strength.

Point constraints may be entered by means of the model editor. Each point in the grid has an associated constraint parameter, which is a real number consisting of an integer part and a fractional part.

The integer part designates the type of constraint to apply as follows:

- (a) 0 means the velocity is unconstrained and can be freely adjusted by GeoTomCG to obtain the best possible agreement with the travel-time data;
- (b) a negative number indicates that the velocity is to be held fixed at the value in the starting model;
- (c) a positive number assigned to a selected group of points indicates that uniformity of velocity is to be maintained over that group by averaging velocities over the group at the end of each iteration. It is useful, for example, for producing uniform layers in the reconstruction. Different groups may be selected and assigned different positive constraint parameters; uniformity is then

enforced for each group independently of the other groups. Only points with raypaths in adjoining voxels are included in the group constraint calculations.

The fractional part of the constraint parameter represents the uncertainty of the constraint, or the fuzz factor  $f$  ( $0 < f < 1$ ). After each iteration, two different velocity values are calculated for each grid point: an unconstrained velocity  $v_0$  determined by the SIRT equations; and a constrained velocity  $v_1$  calculated by strict enforcement of the constraint. The weighted constraints are applied by replacing the velocity from the previous iteration at each point with a linear combination of the unconstrained and fully constrained values ( $v = v_0 * f + v_1 * (1 - f)$  for  $f > 0$ ). Thus larger  $f$  values correspond to greater uncertainty (less regularization) by the constraint.

#### Entire Grid

The whole-grid constraints are upper and lower bounds on calculated velocities and regularization to the average velocity. These constraints are entered in the dialog box for inversion parameters as explained in the following chapter "Performing Inversion Calculations" in section "Select Inversion Options."

Regularization can reduce the velocity variation. Regularization is discussed in many books on tomography. This author has found the discussion by Santamarina and Fratta (1998, pp. 212-223) most helpful. Regularization to the average velocity calculated in the previous iteration can be implemented, but is often not needed because the grid-point basis for velocities creates inherent smoothing as averaging is performed over the voxels sharing each point.

### **Reflecting Plane (Only For Reflected Rays)**

Using reflected rays in addition to direct arrivals can reduce horizontal smearing and improve the reliability of results (Tronicke et al., 2001). If reflected rays are used, then a file giving the position of the reflecting plane must be provided. The reflector plane must be above or below the sources and receivers for the reflected rays, and is specified by three points.

The default file extension is \*.ref, and the example in Samples is called Reflect.ref. This file contains only four lines.

Line 1 is a header for any description.

Line 2 contains the X0, Y0, Z0 coordinates for the first point defining the reflection plane.

Line 3 contains X1, Y1, Z1 for the second point.

Line 4 contains X2, Y2, Z2 for the third point.

The points must be selected so  $Y_0=Y_1$  and  $X_0=X_2$ . Convenient choices are to make point 1 at the X,Y origin, point 2 at the end of the X-axis, and point 3 at the end of the Y-axis. The position of the reflector plane is shown in the display of the model (see “Displaying Tomograms”). If no reflector file is read when there are reflected rays (negative ray identification numbers), results will be meaningless and the program may crash.

## PERFORMING INVERSION CALCULATIONS

To perform the tomographic inversion, follow these steps in this order.

### Open Data File

Click **Data** to view a list of data file types. The format for each type is described in the previous section titled “Data Files (Source and Receiver Coordinates, Traveltimes).” Click the appropriate file type in the list. A standard Windows file selection dialog box will appear. Select the data file you want. The first 1000 lines of the file will be shown to allow you to check if it is the correct file and was read correctly. A display of only 1000 lines does not mean only 1000 lines were read.

### Open Initial Velocity Model (Grid-point) File

Open a model grid-point file for the initial velocity (or 1/attenuation) estimate, prepared as explained in the previous section titled “Initial Velocity Model.” A velocity model file generated by this GeoTomCG has the extension 3mq. For backward compatibility, model files in the previous 3dm format without Data quality for points and Slowness residual/slowness can be read. A model grid-point file must be opened or GeoTomCG will not run. Click **Model, Open model file**. A standard Windows file selection dialog box allows you to select the file. The first 1000 lines of the file will be shown. GeoTomCG reads and writes the number of raypaths in each voxel and the anisotropy for each point. Initial values of Data quality for points and Slowness residual/slowness are not used in the inversion and are set to 1 during generation of the initial model.

### Select Inversion Options

Select options for running by clicking **Options, Inversion parameters**. The dialog box shown to the right will appear.

#### Iteration and Velocity Limits

The top two text boxes allow you to enter the number of straight-ray iterations and the number of curved-ray iterations by ray-bending. While the program is running, you can stop program execution after the next iteration by pressing the n or N key.

The next set of text boxes is for entering the number of wavefront iterations and the time step, if wavefront iterations are used. The ray-bending method is recommended for most data sets, and you will usually leave the number of wavefront iterations at 0. See the chapter "Wavefront Construction" for information on this method.

The buttons in the box labeled **Velocity correction weighting method** allow you to choose the weighting method used to apply residuals to correct velocities at grid points during the inversion. **Raypath fractions per point** uses the sum of the fractions of raypaths influencing each grid point for those weighting factors. This method makes long and short raypaths have the same total influence on the calculated velocity pattern. The influence of shorter raypaths is concentrated in fewer points and the influence of longer raypaths is spread over more points. This method is better for most data sets and is recommended unless there is a special reason to use the other one. **Raypath segment lengths per point** uses the sum of the lengths of raypath segments as those weighting factors. Longer raypaths then have more total influence. This method may be better if the strong influence of short raypaths on the grid points they pass near is undesired. One example is a set of source and receivers around a column with some sources and receivers close together and having too much influence on the grid points between the closely spaced source and receiver positions.

The bottom two text boxes allow you to enter minimum and maximum allowed velocities. The default minimum allowed velocity is half of the smallest straight-line velocity for the data file, and the default maximum is double the largest straight-line velocity. You may want to specify a smaller maximum or a larger minimum velocity. For example, radar velocity in rock is always less than the velocity in air. Specifying tighter limits helps to counteract nonuniqueness and avoid unrealistic results. Enter the desired values in the text boxes, and click **OK**.

If desired, the user may set smoothing or damping parameters or anisotropy type by clicking the corresponding buttons before clicking **OK**.

### Smoothing

Smoothing consists of replacing the velocity at a point by a weighted average of the velocity at that point and the velocities at neighboring points. It can be done during inversions at each iteration, or performed on a velocity model file after inversions are completed. To perform smoothing during inversion at each iteration, click **Smoothing** in the **Options, Inversion parameters** dialog box. A dialog box is then displayed that allows the user to select smoothing in all directions, in the horizontal direction, or to assign the weighting to individual neighboring points. After setting the values as desired, click **OK** to return to the **Inversion parameters** dialog box.

To perform smoothing of an existing model, open a model, then click **Model, Smooth**. A dialog box identical to the one for smoothing during inversion will appear. After setting the values as desired, click **OK** and a standard dialog box for selecting the name of the smoothed model file will appear.

Note that points without raypaths are not used in any way in the smoothing. They are not smoothed, and their weighting is not used in smoothing other points. Points with no raypaths are excluded so that points with no information about the calculated velocity pattern do not degrade the smoothing. This also means that a newly generated velocity model with no raypaths listed in the voxels will not be smoothed.

When some points are not used because of no raypaths or when some neighboring points are not available because the point being smoothed is on a surface, then the weighting for the center point is increased to make the sum of the weights equal 1.

### Damping

Damping is regularization to the average velocity calculated after each iteration. For the first iteration, the average velocity is the average straight-line velocity of all raypaths. For subsequent iterations, the average velocity is the average for all points that are affected by raypaths. Points for zones without raypaths are not used in those calculations.

To perform damping, click **Damping** in the **Options, Inversion parameters** dialog box, then enter a value for the damping coefficient in the text box. After setting the value, click **OK** to return to the **Inversion parameters** dialog box. Increasing the value of the damping coefficient increases the effect of the average velocity and decreases the point-to-point variation in the velocity. A damping coefficient that is too large will do more harm than good, as it will force too much uniformity and hide important velocity anomalies. If data are accurate, it is usually not needed. The grid-point basis for velocities creates inherent smoothing as averaging is performed over the voxels sharing each point. If damping is used, start with a small coefficient, for example 0.01, then gradually increase it to determine the effect on the residuals

### Anisotropy Type

The user can set the appropriate Thomsen anisotropy parameters (Thomsen, 1986; Thomsen, 2002) for polar anisotropy (often called transverse isotropy). Click **Anisotropy** in the **Options, Inversion parameters** dialog box, then select the appropriate type of anisotropy, P-wave, parallel S-wave, or transverse S-wave.

For P-waves, the user can enter a value for the departure from ellipticity that is the Thomsen parameter  $\eta$ . If  $\eta = 0$ , the anisotropy is elliptical. If  $\eta$  is positive, the velocity for raypaths near  $45^\circ$  from the axis of anisotropy is smaller than it would be for elliptical anisotropy.

Parallel S-wave anisotropy means that the vibration direction of the S-waves is parallel to the plane of symmetry, which is perpendicular to the axis of anisotropy. An example is the common S-wave source of a weighted plank lying on the ground struck on the end with a hammer, over rock with horizontal layering. Conversely, transverse S-wave anisotropy means that the vibration direction of the S-waves is perpendicular to the symmetry plane, which is parallel to the axis of anisotropy. An example is a downhole hammer sliding vertically to generate vertical S-waves in rock with horizontal layers.

For parallel S-waves,  $\eta$  is always 0 for polar anisotropy.

For transverse S-waves, the user can enter a value for  $\eta$ . This form of anisotropy also depends on the ratio P-wave velocity/S-wave velocity. The user can enter a typical value. For polar transverse S-wave anisotropy, the velocity parallel to the axis of anisotropy is the same as



the velocity perpendicular to it. The maximum velocity occurs for raypaths at  $45^\circ$  to the anisotropy direction.

### **Specify Raypath File Name (Optional)**

If you want to save raypaths, click **Raypaths, Save Raypaths**. You are asked to provide a name for the raypath \*.ray file to be saved. A raypath file contains the 3D coordinates of raypaths that can be read and shown while displaying the model as described below in “Displaying Tomograms and Saving Output Files.”

### **Specify Reflector File (Only For Reflected Rays)**

If desired, include a reflector. As explained above in “Generating and Editing Input Files,” GeoTomCG provides an option for including rays reflected from a known plane. Click **Reflectors, Open reflector file**. The default extension is \*.ref. The position of the reflector plane is shown in the display of the model (see “Displaying Tomograms”). If no reflector file is read when there are reflected rays (negative ray identification numbers), results will be meaningless and the program may crash.

### **Start Inversion**

Start the inversion by clicking **Run** on the main menu. Two options will be displayed, **Run standard inversion** and **Run curved-ray attenuation**. You will usually click **Run standard inversion**. To indicate progress, the program displays the ray identifier for every hundredth raypath completed and the RMS residual and sum of residuals for each completed iteration. A message box indicates when the program execution is complete.

Use **Run curved-ray attenuation** for curved-ray amplitude inversions if you have a raypath file from a travelttime inversion. This option uses the coordinates in a raypath file from a travelttime inversion as fixed raypath coordinates during the amplitude inversion to calculate attenuation coefficients. An amplitude inversion cannot be used to generate reliable curved rays because ray bending is determined by changes in velocity. GeoTomCG will first ask for the name of a raypath file to read, then the rest of the program execution will be like a standard inversion. The raypaths in the amplitude inversion must correspond exactly with the raypaths in the travelttime inversion. If any raypaths are omitted from the amplitude inversion that were included in the travelttime inversion, the wrong raypaths will be used with the amplitudes following the omitted raypaths. The travelttime inversion can be redone omitting whatever raypaths are omitted from the amplitude inversion to obtain matching raypaths.

## OUTPUT FILES GENERATED AUTOMATICALLY

GeoTomCG generates the following three types of output files automatically during inversions.

### Calculated Velocity Model File

This version of GeoTomCG generates a model file with default extension 3mq each iteration. Previous versions generated model files with the extension 3dm. For backward compatibility, GeoTomCG can read \*.3dm files, but it writes \*.3mq files. The \*.3mq files contain the 3D coordinates and calculated velocity (or 1/attenuation coefficient) for each point in the format of the input velocity model plus additional columns with quality factors in the following format: X Y Z Velocity Constraint Anisotropy Data quality Slowness residual/slowness. Data quality and Slowness residual/slowness are calculated during the inversion; their initial values are not used in the inversion. Data quality for each grid point is calculated using the Data quality for raypaths. The methods for calculating Data quality and Slowness residual/slowness are described in the next section “DISPLAYING TOMOGRAMS AND SAVING OUTPUT FILES.” The file is generated each iteration so you can stop execution before the final iteration and have results. The previous file is overwritten each iteration. You are asked to provide a name for the model \*.3mq file after clicking the menu options **Run**, then **Run standard inversion** or **Run curved-ray attenuation**.

### Residuals File

A residuals file with default extension rsd is generated after the final iteration. It contains the RMS residual for each iteration and the residual for each raypath for the final iteration. The residual equals measured time minus calculated time. You are asked to provide a name for the residuals \*.rsd file after clicking the menu options **Run**, then **Run standard inversion** or **Run curved-ray attenuation** and providing a name for the \*.3mq model file.

### Raypath File

A raypath file with default extension ray is generated if you select **Raypaths, Save raypaths**. A raypath file contains the 3D coordinates of raypaths that can be read and shown while displaying the model as described in “DISPLAYING TOMOGRAMS AND SAVING OUTPUT FILES” below.

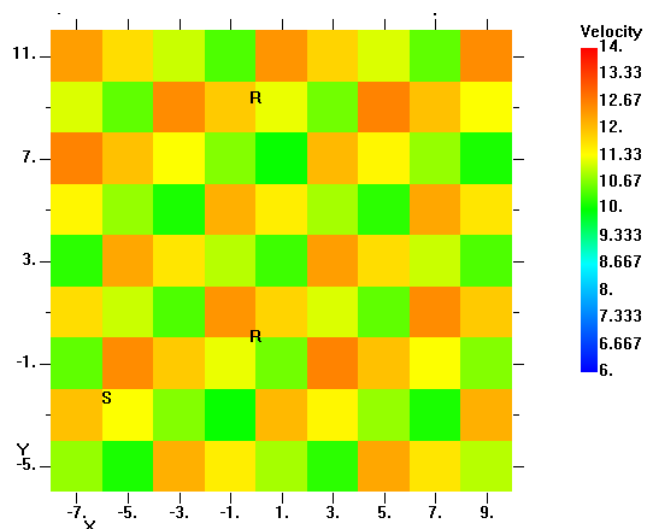
## DISPLAYING TOMOGRAMS AND SAVING OUTPUT FILES

A tomogram is a 2D or 3D representation of the calculated velocities or 1/attenuation coefficients. GeoTomCG displays tomograms with colors in small rectangles representing the velocity (or 1/attenuation coefficient after amplitude inversions) at the point in the center of each rectangle.

After loading a model file, select **Model, Display model**. The dialog box shown below will appear in the upper left corner of the screen. Select a 2D or 3D view with the upper radio buttons.

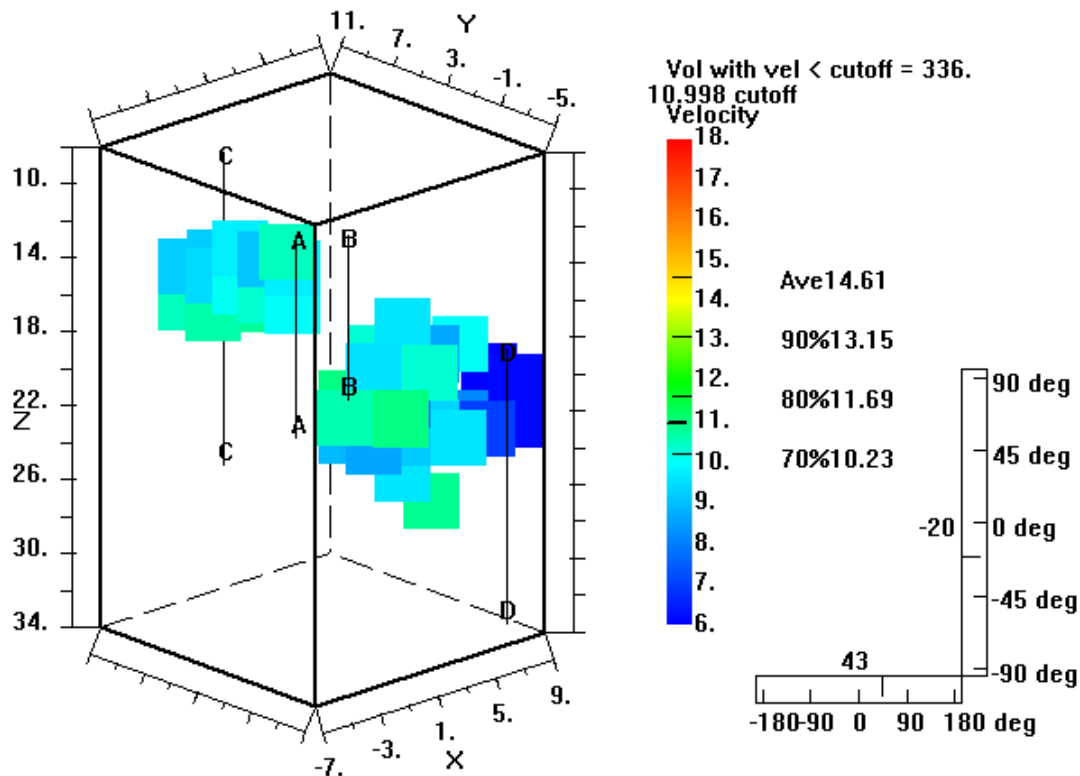
If you select a 2D slice plane, then select the slice number with the up-down control, which also indicates the slice position in the plane. Select what parameter to display. Click **Show slice**. The slice will be shown in a scrollable window. The source and receiver positions from the open data file are shown as S and R, respectively. The title at the top of a 2D display shows the slice number and position. The slice number can be changed with the **Page Up** and **Page Down** keys while viewing the tomogram.

The **Tilted** view is a new display option. It is not a diagonal view, but rather shows the velocity at the grid points closest to the tilted planes described in the section Editing the Model, Applying. It can have the appearance of a multi-colored checkerboard especially with the large voxels in Demo1.3dm, as the velocity at neighboring grid points can be closest to different option layers. This tilted display is for the Demo1.3dm editing example in Applying 3D Tilted Layers with point 1 at Z = 16.



An option provides a simplified color spectrum with green indicating fast and red to magenta indicating slow with yellow between green and red. This spectrum was requested for processing data from concrete pilings. The conventional red for fast and blue for slow with a full spectrum between is still the default. The user can select the green to red spectrum with one click by clicking on the **Green Red** button to the right of the **Adjust display colors** button and can return to the red to blue spectrum by clicking on the **Red Blue** button. The **Adjust display colors** button allows you to select other spectra or create and save a custom spectrum.

If the user selects **3D Rect**, the display is like that shown below. The data are the Demo1WithLabels.3dd data set included with the GeoTomCG program.



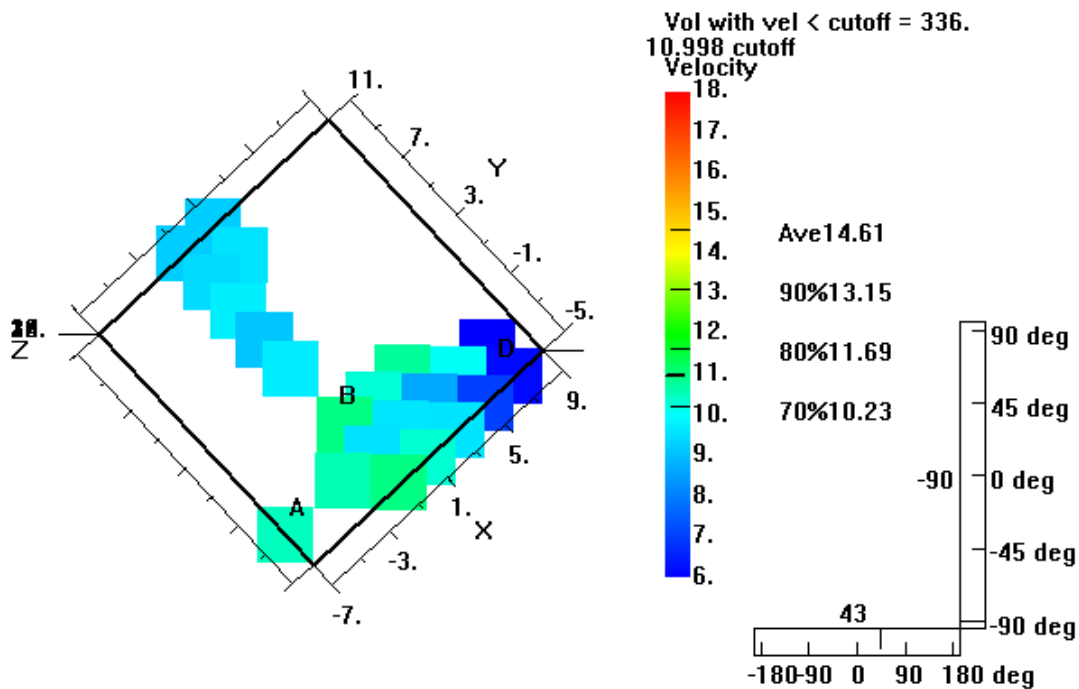
In this display, red represents the highest velocity and blue the lowest. Often the position of low-velocity zones is of most importance, so in this display a low cutoff velocity is selected so only the low-velocity zones are colored. The average velocity and the velocities corresponding to 90, 80, and 70% of the average are shown beside the color key, and are indicated by light horizontal lines in the key. The average velocity is the average over all raypaths of source to receiver distance/traveltime. If no raypaths have been read, then the average is calculated for all grid points with more raypaths than the cutoff for **Rays per point**. **Rays per point** is the number of rays that affect the calculated velocity at that point, which is the sum of the rays in voxels with a corner at that point.

Clicking on the color key selects a cutoff velocity for the display to reveal zones of low velocity. The example shows a cutoff of 10.998, indicated by the heavy horizontal line in the key. The volume surrounding a node with velocity greater than or equal to the cutoff velocity is

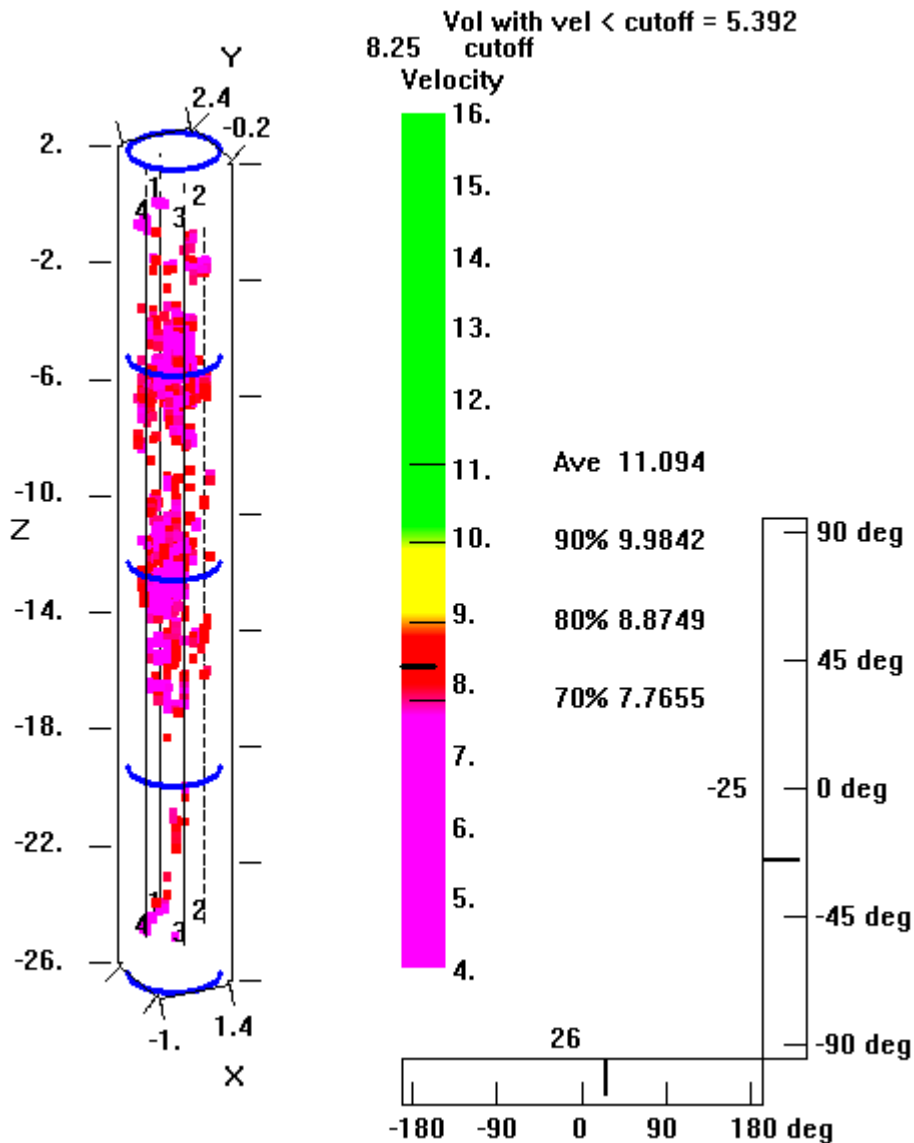
not colored in the display. Nodes without raypaths are not displayed in any case. The outline of the outermost grid points is drawn to help show the 3D perspective. The viewer can rotate the display to help show on which side of the display a region of interest is located. Boreholes are indicated by thin vertical lines. The ray density cutoff for displaying colors around grid points can be selected by the user and entered in the edit box labeled **Rays per point** at the lower left of the dialog box.

The angle of rotation can be selected by clicking on the horizontal box. Zero deg corresponds to the X axis in front and back. The Y axis is at the front and back when the rotation angle is 90 deg. The angle of tilt can be selected by clicking on the vertical box. The Y axis labels are displayed at the top for small tilt angles to avoid overlap with the X axis labels.

The tilt of the display can be rotated as in the display below to show the horizontal position of the low-velocity zones.



If the user selects **3D Cyl**, the display is like that shown below. The data file was provided by Olson Engineering. Olson Engineering also offered many suggestions to make the display more useful for examining concrete piles. Using those comments, the 3D cylinder display was designed to be helpful in quickly viewing zones of low velocity. In this display, green represents the highest velocity and red to magenta the lowest, following the recommendation of Olson Engineering. Similar to the 3D rectangle display, the average velocity and the velocities corresponding to 90, 80, and 70% of the average are shown beside the color key, and are indicated by light horizontal lines in the key. Clicking on the color key selects a cutoff velocity for the display to reveal zones of low velocity. The example shows a cutoff of 8.25, indicated by the heavy horizontal line in the key. The volume surrounding a node with velocity greater than or equal to the cutoff velocity is not colored in the display, nor are nodes without raypaths. The borehole labels 1, 2, 3, and 4 are shown at the top and bottom source and receiver positions. The arcs help to show perspective, and do not have any other significance.



The volume with a velocity less than the cutoff is shown to the right of the cutoff velocity. The volume is calculated as (number of nodes with velocity < cutoff velocity) x (volume for each voxel). This result includes non-contiguous nodes with calculated low velocity, and so may overestimate the true volume of void space that needs remediation.

As in the 3D rectangle display, the angle of rotation is selected by clicking on the horizontal box. Zero deg corresponds to the X axis in front and back. The Y axis is at the front and back when the rotation angle is 90 deg. The angle of tilt is selected by clicking on the vertical box. The Y axis labels are displayed at the top for small tilt angles to avoid overlap with the X axis labels. The 3DCylSynth data set provided with GeoTomCG can be used for trying these and other display options. 3DCylSynth2Defects.3dd is a synthetic data set simulating a concrete column with a low-velocity zone on the surface at  $X = Y = 0$  m,  $Z = 4.75$  to  $5.25$  m, and another one near the center at  $Z = 14.75$  to  $15.25$  m. 3DCylSynth2Defects\_1Str7Bend.3dm is the calculated velocity model file using 3DCylSynthUnif3.3dm as the initial model file.

Source and receiver positions are shown by thin vertical lines in the two 3D displays rather than the S and R used in 2D displays. The number of source and receiver positions is often so large when examining concrete pilings that the S and R letters would cover much of the display.

You have the choice under **Show sources and receivers?** to show:

- a) **All** all sources and receivers are displayed,
- b) **In plane** in 2D slices, sources and receivers in the selected plane are displayed  
in 3D views, sources and receivers not in back of colored voxels are displayed.
- c) **None** no sources or receivers are displayed.

**In plane** is the default. If any of the three 2D displays are shown and **In plane** is selected, the sources and receivers no more than half the corresponding voxel dimension from the corresponding plane are shown. For example, if the XZ plane is shown, the sources and receivers with the Y coordinate within half the voxel Y dimension are displayed. If either of the two 3D views (3D rectangle or 3D cylinder) are shown and **In plane** is selected, then the sources and receivers in the front half of the display are drawn after the colored voxels so they can be seen. The sources and receivers in the back half of the display are drawn before the colored voxels, so they may be covered by the voxels unless a low cutoff velocity is chosen. The positive Y axis points back when the rotation angle is near 0. The hiding of sources and receivers in the back half of the display helps the user view the perspective and identify the positions of low-velocity zones. Parts of the source and receiver positions showing the boreholes are hidden in the 3D displays above.

**Rays per point** (or **Rays per grid point**) is the number of rays that affect the calculated velocity at that point, which is the sum of the rays in voxels with a corner at that point. Thus a point with surrounding voxels that contain rays can have more rays per point than the number of rays in any one of those voxels. In 2D displays, grid-points with fewer raypaths than the cutoff number in the text box labeled **Rays per point** are shown with a hatched pattern around that point. The default cutoff value for **Rays per point** is 8% of the maximum number of rays per point. Data sets with a high ray density in part of the volume of investigation and a much lower ray density in other parts may require a lower **Rays per point** cutoff value to display properly the calculated velocities in all parts of the investigated volume.

Raypaths can be overlaid if a raypath file is read by clicking **Raypaths, Read raypaths**. Rays are shown for all slices, so rays may appear where the hatched pattern indicates no raypaths in that slice. While raypaths are displayed, right-clicking on a source position limits the displayed rays to those from that source. Then left-clicking on a receiver position will show the ray identification and measured and calculated times. To resume displaying all rays, click **Raypaths, Resume raypath display**. The cursor must be no more than  $\frac{1}{4}$  of a voxel width or height away from the source or receiver position to select it. Increasing the horizontal and/or vertical magnification, as explained in the following paragraph, can facilitate the selection.

The size of the display in the horizontal and vertical directions can be adjusted by clicking on **Model, Display Model**, then entering new values for the horizontal and vertical magnification factors in the lower left text boxes. This feature can provide vertical or horizontal exaggeration by using different values for the horizontal and vertical magnification factors.

If desired, lines represented in a file can be overlain on the display. The format for this file is the same as the Surfer BLN files, except that the values must be separated by spaces instead of commas. The format is:

```
npts  flag  (npts is the number of points in first overlain line, flag is not used here.)
x1   y1    (coordinates of first point in this line.)
x2   y2    (coordinates of second point in this line.)
xn   yn    (coordinates of last point in this line, must be identical to first point.)
npts  flag  (number of pts in next overlain line, flag is not used.)
x1   y1    (coordinates of first point in second line.)
```

continuing in the same way for all the lines to be overlain on the display.

The overlain lines can represent topographic lines or anything that the user wants to draw on the display. After preparing the file, give it a BLN extension because it is like the Surfer BLN blanking file. Display the model as usual, being sure that the display is in the same plane as the BLN file. Then select the **Overlay Surfer BLN file?** near the lower right in the **Model, Display model** dialog box and click **Show slice**. The program will display a standard **Open file** dialog box for entering the name of the BLN file. Enter the name of the file and click **OK**. The file will be opened and the lines overlain on the display. The user is responsible for making sure that the appropriate plane is being used for the model display. The lines will continue to be overlain as the slice number is changed. To stop overlaying the lines, deselect the check box for **Overlay Surfer BLN file?**

If the check box is deselected when **Show slice** is clicked, the BLN file is closed. The next time the check box is selected, the user can enter the name of the BLN file. Therefore, to overlay lines from a different BLN file, click **Model, Display model**, deselect the **Overlay Surfer BLN file?** check box, and click **Show slice** to show the model without overlain lines. Then again click **Model, Display model**, and select the check box. When you click **Show slice**, you will be asked to enter the name of the BLN file.

The user can adjust the color spectrum. The **Model, Display model** dialog box has a button called **Adjust display colors**. Click it to open a dialog box for adjusting the colors. Several named spectra can be selected by double-clicking the list, or you can adjust individual RGB



values in Edit boxes. The option can read and save Surfer color map files, but is limited to 7 anchor points. If the 100% anchor point is before point 7, the points to the right of the 100% point are ignored. RGB values in points to the right of the 100% point are set to 0 and ignored when reading color maps that have fewer than 7 anchor points. When you are satisfied with the spectrum colors, click **OK** to go back to the **Model, Display model** dialog box.

Parameters other than the velocity can be displayed by selecting the radio buttons **Rays per grid point**, **Constraints for points**, **Anisotropy for points**, **Data quality for points**, or **Slowness residual/slowness** instead of the default **Velocity at grid points**. In 3D views, the same cutoff velocity for the display will be in effect while viewing the other parameters. Thus, if you have selected a low cutoff velocity and then view **Rays per grid point**, the only voxels that will be colored are the same few that were colored in the **Velocity at grid points** display. The **Rays per grid point** display can help determine if the default cutoff for **Rays per point** is appropriate. The **Constraints for points** and **Anisotropy for points** displays can help determine if editing of the model to set those parameters to different values in different zones was done correctly.

**Data quality for points** is the average of the raypath data quality weighting for the raypaths that affect the calculated velocity at the point. The minimum value is 0 and occurs if no raypaths are close enough to affect velocity at that point. The maximum value is 1 and occurs if all raypaths affecting the velocity have data quality weighting = 1.0. So, the grid point data quality =  $\text{Sum}[(\text{raypath data quality})]/\text{number of raypaths}$ , with the sum taken over all raypaths affecting the velocity for that grid point. If all the raypaths in the sum have the same raypath data quality, then grid point data quality = raypath data quality.

**Slowness residual/slowness** is a dimensionless parameter indicating the goodness of fit for raypaths affecting the velocity at that point. Slowness is the reciprocal of velocity. Good fits will have values less than 0.1. It is calculated from the following expressions:

The residual for each raypath = measured time – calculated time.

The slowness residual for each raypath = residual/raypath length.

The slowness residual for each voxel =  $\text{Sum}[(\text{slowness residual for each ray segment in the voxel})]/(\text{number of ray segments in the voxel})$ .

The slowness residual for a grid point =  $\text{Sum}[(\text{slowness residual for each voxel with a corner at that point})]/(\text{number of voxels with a corner at that point})$ .

Then Slowness residual/slowness for each grid point =  $[(\text{slowness residual for that grid point})/(\text{calculated slowness for that grid point})]$  or, equivalently,

Slowness residual/slowness = (slowness residual for that grid point) \* (calculated velocity at that grid point).

### Copying To Windows Clipboard

The user can copy selected areas of any of these plots or the model display to the clipboard for pasting into other Windows application. Press the left mouse button at the upper left corner of a desired bounding rectangle. Then drag the mouse, with the left button still down, to the lower right corner of the desired bounding rectangle, in the usual manner for selecting an area for copying. Then click **Edit, Copy** and the bitmap inside the bounding rectangle will be copied to the clipboard.

Raypaths will disappear when you press the left mouse button for drawing the rectangle, as the screen is refreshed. To include raypaths in the copying, resume displaying rays by clicking **Raypaths, Resume raypath display**. The bounding rectangle will disappear, but the program has stored its locations. When you click **Edit, Copy**, the rectangle will reappear and the bitmap inside the rectangle, including the raypaths, will be copied to the clipboard.

### **Slice File, Slice Aligned with Coordinate Axes**

A slice file in one of the coordinate planes can be saved. These files contain 2D point coordinates in the selected plane, the velocity, the number of raypaths affecting that point, grid point data quality, and slowness residual/slowness. These files can be read by Surfer and, with changes in the header lines, by other 2D contouring programs.

A slice file can be saved while displaying a tomogram. After opening a model file, click **Model, Display model**, and select the plane and slice number. If **Tilted** is selected, the saved file will contain the parameters for the displayed grid points. Slice files cannot be saved for the 3D views. When you are displaying the slice for which you want to save a file, click **Save slice file while showing slice**, then click **Show slice** again. A standard **Save As** file dialog box will open. The user can enter a name and click **Save** or can click **Cancel** to not save the file.

### **Diagonal Slice File**

Diagonal slices (vertical or dipping) can be generated. A vertical diagonal slice file is generated from the **Model, Extract vertical slice** menu. It extracts a vertical slice over the height of the grid from a vertical line A to a second vertical line B. The user must enter the values for the X and Y coordinates for the two lines in a dialog box. A **Save As** dialog box allows the user to enter the name of a file to be generated.

The output for diagonal vertical slices contains the distance from line A, the Z coordinate, the velocity (or 1/attenuation) plus the quality parameters Interpolated Rays/Point, Interpolated Data Quality, and SlownessResid/Slowness for the points in the vertical slice file. These parameters are calculated by linear interpolation of the values at the four grid points that surround each of the vertical slice points, except that a grid point having no raypaths is not included in the interpolation. The Z coordinate in the vertical slice matches the Z of the grid point, but the vertical slice plane can be diagonal so the X and Y coordinates for grid points do not match the horizontal distance from line A. Thus, each vertical slice point is between four grid points. The interpolation uses the distance from each of the four grid points to the vertical slice point. Click **Cancel** to clear the dialog box without saving.

A diagonal slice that has a horizontal X axis and dips in the YZ plane is generated from the **Model, Extract slice, hor X axis** menu. The user must specify the Y and Z coordinates for the two lines A and B parallel to the X axis. In the same manner, a diagonal slice that has a horizontal Y axis and dips in the XZ plane is generated from the **Model, Extract slice, hor Y axis** menu. The user must specify the X and Z coordinates for the lines A and B parallel to the Y axis. These files contain for each point in it the distance from line A, the Z coordinate, the

velocity (or 1/attenuation after amplitude inversions), plus the quality parameters Interpolated Rays/Point, Interpolated Data Quality, and SlownessResid/Slowness.

### **Exported Files for 3D Graphics Programs**

The user can export files of 3D data for input to 3D graphics programs with two different formats, one for Tecplot, Voxler and most other 3D graphics programs and the other for Slicer Dicer.

#### Tecplot - Voxler Format

Tecplot is sold by Amtec Engineering, Inc., website [www.amtec.com](http://www.amtec.com). Voxler is sold by Golden Software, Inc., website [www.goldensoftware.com](http://www.goldensoftware.com). Other 3D graphical display programs that read data in this format are also available.

After opening the desired model file, click **Model, Export 3D Tecplot or Voxler file**, and enter a name for the file to be exported. The top several lines of the resulting file contain display control parameters read by Tecplot. Other lines starting with # are comment lines containing information that may be needed by other graphics programs. Voxler allows the user to skip those lines and start input with the lines containing data. There is one line for each point of the model file, with the following format:

X Y Z Velocity Constraint Anisotropy Rays/point Point data quality SlownessResid/Slowness

The Rays/point is the number of raypaths in all of the voxels that share that grid point, so it can be surprisingly high. The sum of the rays/point is greater than the number of raypaths. The rays/point can be used by Tecplot for blanking (leaving uncolored) points where there are no raypaths. Voxler and most other 3D graphical programs do not have that capability, but allow the display to be transparent for specified velocities. GeoTomCG provides an option for all grid points with raypaths less than a user-specified number to be set to a user-specified low value so the display can be made transparent for those grid points.

The next lines in the file 3D Tecplot file list the source and receiver positions in a format that Tecplot uses to generate 3D lines, which allows the source and receiver boreholes to be displayed. These lines are initially commented out by #, so delete the # if you want Tecplot to show the borehole positions. Tecplot can also read the raypath file saved by GeoTomCG.

#### Slicer Dicer Format

Slicer Dicer is sold by Pixotec, LLC, website [slicerdicer.com](http://slicerdicer.com). The format below is the one used when this export option was added to GeoTomCG at the request of a user. The website indicates that additional input formats are available now.

After opening the desired model file, click **Model, Export 3D SlicerDicer File**. You will see a dialog box titled **Set velocities for points with few raypaths** with text boxes **Set velocity =** (default 1) **at nodes with raypaths <** (default 1). This option was added at the request of a Slicer Dicer user because Slicer Dicer does not allow blanking of the velocities based on the raypath coverage. It is important to distinguish between velocities calculated by the inversion

and velocities remaining at the initial value because of no raypaths there. The velocity at grid points where there are no raypaths is not meaningful, and should not be displayed in the same way as the calculated velocities.

The method devised for Slicer Dicer users to distinguish the calculated velocities from the uncalculated velocities was to add an option to reset the uncalculated velocities in the Slicer Dicer exported file to some low value so they could be identified in the Slicer Dicer contoured display. You may want to choose a velocity lower than 1 if your inversion calculates velocities less than or close to 1. You may require more than one raypath as a criterion for displaying a velocity as a reliable calculated velocity. If you do not want any of the velocities to be reset by this option, enter 0 in the text box for **at nodes with raypaths <**.

After clicking **OK**, a dialog box for selecting a name for the exported file will be shown. Enter or select a file name and click **Save**. The top several lines of the resulting file contain parameters needed by Slicer Dicer. Then one line for each grid point of the model file follows, with only the point velocity listed. The coordinates of each point are not written because Slicer Dicer does not read that format.

The option **Model, Reset velocities where few rays** below **Export 3D SlicerDicer File** allows the user to generate a GeoTomCG model file in the standard 3mq format, replacing velocities for grid points with less than the specified raypaths by the specified velocity. This option may be useful for 3D graphics display programs that cannot perform blanking based on raypath coverage.

### **Exported 2D Raypath File for Surfer**

The user can export a file of 2D raypath coordinates for Surfer or similar 2D graphics programs as a comma-delimited file used by Surfer for blanking or showing lines. If raypaths are displayed, click on **Raypaths, Export 2D ray file**. It will read the opened 3D raypath file and form a 2D file by leaving out the coordinate appropriate for the slice direction. If raypaths are not displayed, it provides a dialog box to select the 3D raypath file to read. It then provides a dialog box to name a \*.BLN file for saving the 2D raypaths.

## PLOTS AND FILES FROM RESIDUALS FILES

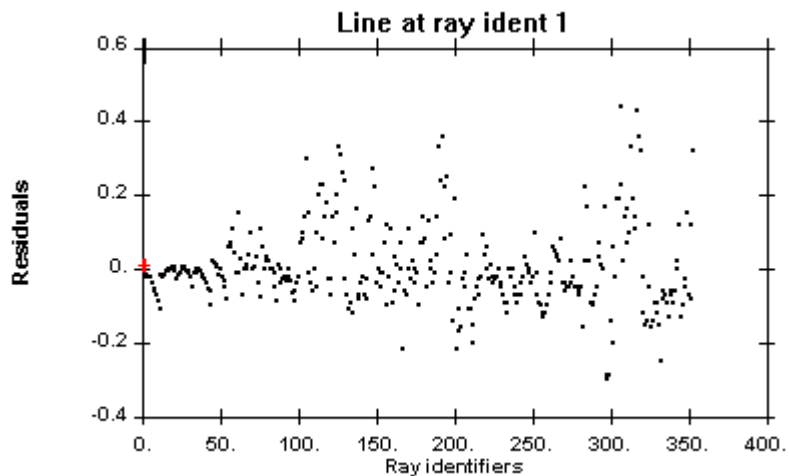
### Plotting Variables

The variables in residuals (\*.rsd) files can be plotted. Open a residuals file by clicking **Residuals, Open residuals file**. Then click **Residuals, Plot**, and this dialog box with plotting options will appear in the upper left corner of the screen.

Normalized residual is the residual divided by the measured time.

The plot from the settings in the dialog box using data from Demo1 is shown below it.

When the variable on the horizontal axis is the ray identifier, the user can determine the ray identifier for a particular point by right-clicking on or just to the right of the point and, if necessary, using the left and right arrow keys to move the indicator by one unit. The small line at the top shows the approximate position and the caption at the top shows the exact identifier. These are ray identifiers, not the ray subscripts in the array, so the user can assign meaningful ray identifiers and select different sections of the residual files for plotting.



The option for plotting ray angle versus velocity is to help reveal possible anisotropy or layering. If layering is present, the maximum velocity is usually in line with the layering. The ray angle is in degrees from a horizontal (XY) plane for a straight ray traveling directly from source to receiver. A positive angle means the source-to-receiver path is upwards. If the default setting of **Down** was used under **Options, Pos Z axis up or down** when the residuals file was generated, then upward corresponds to decreasing Z (e.g. depth). Conversely, if **Up** was specified under **Options, Pos Z axis up or down** when the residuals file was generated, then upward corresponds to increasing Z (e.g. elevation) and a positive angle indicates that the receiver Z is greater than the source Z. The plot uses the angles in the residuals file, so the sign of angles does not change if the setting of **Pos Z axis up or down** is changed while the plot is being viewed.

### **File of Calculated Times in Data File Format**

The user can generate a 3dd data file containing calculated times. The calculated times are determined by reading a residuals file, then using measured time minus residual as calculated time. Open a residuals file by clicking **Residuals, Open residuals file**. Then click **Generate calculated data file**. A common dialog box for designating the name of the 3dd data file to be saved will appear. Provide a file name, click **OK**, and the data file will be generated. The data file is not loaded automatically. The time-picking program TomTime can read this file and overlay calculated times on the waveform traces to aid in checking time picks.

### **File of Measured Times with Outliers Removed**

The user can generate a new file of measured data with some raypaths removed, using several criteria to remove outliers or other suspect ray paths. Outliers can be removed before inversion using a data file or after inversion using a residuals file. Removing outliers using a data file is explained in the earlier chapter "Generating and Editing Input Files," near the end of section "Data Files (Source and Receiver Coordinates, Traveltimes)."

After inversion, outliers can be removed using the following criteria: ray identifier, straight-line velocity (or 1/atten coefficient), ray angle from horizontal in degrees, residuals, and normalized residuals. Open a residuals file, then click **Residuals, Remove outliers in measured data**. A dialog box will appear for selecting criteria and their ranges. A standard **Save As** dialog box will appear for saving a file of measured data with the selected outliers removed.

## CHECKING FOR AND APPLYING ANISOTROPY

GeoTomCG provides options for examining whether anisotropy may be present, and for applying anisotropy. It does not solve for anisotropy as part of the inversion, so the magnitude of anisotropy is determined by examining the RMS residual as the magnitude is varied over a series of inversions.

The general procedure to check for anisotropy is to use the **Residuals, Plot** option to examine the dependence on ray angle of the straight-line velocity and the normalized time residual. The straight-line velocity is source-to-receiver separation/measured travelttime. Recall that the normalized time residual is (measured time - calculated time)/measured time. A strong dependence on raypath angle and consistency between the two plots would suggest that anisotropy may be present. A significant decrease in RMS residual with the addition of anisotropy would be further evidence. Be cautious when applying anisotropy, however. A dependence of velocity on raypath angle does not necessarily confirm anisotropy. Site conditions other than true anisotropy can create this dependence.

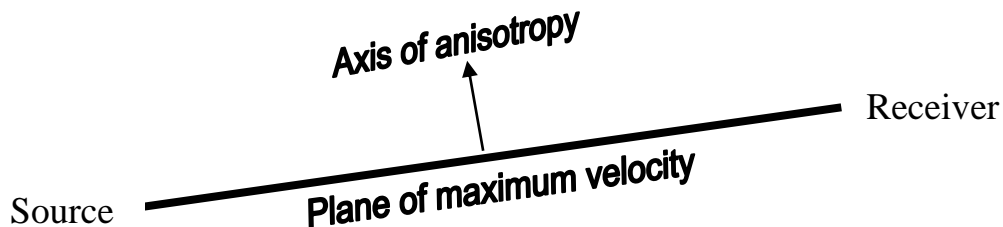
Five illustrated examples will show how to check for anisotropy and how to apply it. The five examples represent:

1. No anisotropy
2. Ambiguous evidence of anisotropy, unconfirmed by residuals.
3. Anisotropy with the maximum velocity in the horizontal plane.
4. Anisotropy with the maximum velocity in a dipping plane.
5. Nonuniform anisotropy and editing the velocity model accordingly.

We assume that the anisotropy is transverse isotropy. Then:

- a) There is a plane of maximum velocity in which the velocity is the same in all directions.
- b) The velocity is a minimum in the direction perpendicular to the plane of maximum velocity for P-waves and parallel S-waves (see "Anisotropy Type, page 33).

In GeoTomCG, the axis of anisotropy is perpendicular to the plane of maximum velocity. The tilt of that axis is measured from the positive direction of the Z coordinate axis, which can be either up or down, depending on the setting under **Options, Pos Z axis up or down**.



### Example 1. No anisotropy

The tutorial data set Demo1 is an example with insignificant evidence for anisotropy. A plot of straight-line velocity versus ray angle is shown to the right.

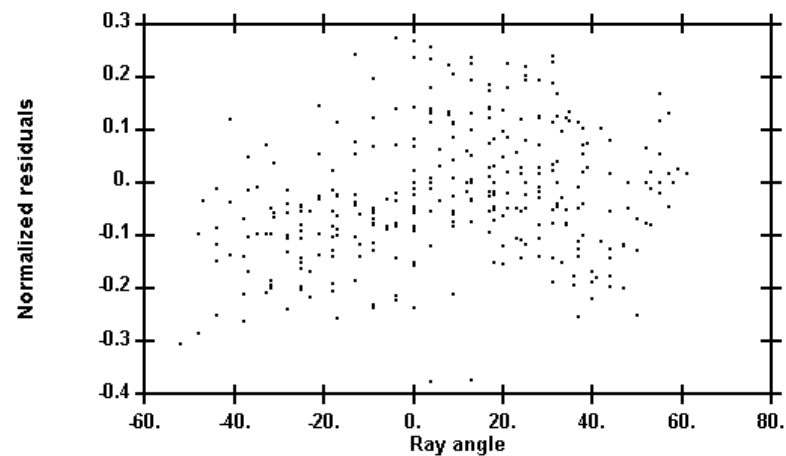
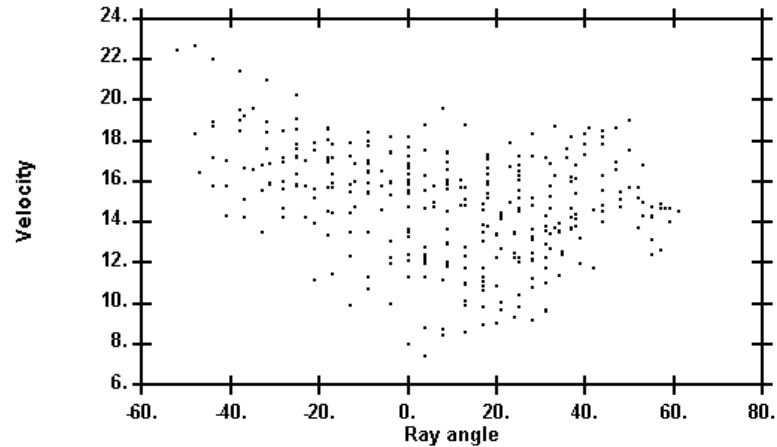
There is no clear pattern of dependence of velocity on ray angle. Similarly, the plot below it does not indicate a significant

dependence of normalized residual on ray angle.

Rows in the velocity model file that control anisotropy are shown below. The anisotropy direction of  $X = 0$ ,  $Y = 0$ , and  $Z = 0$  means that there is no axis of anisotropy, and no anisotropy calculations are performed.

The last two lines shown below are the first two lines of the individual grid-point descriptions. The last column is the anisotropy magnitude at each grid point. Recall that the anisotropy magnitude is the ratio (velocity perpendicular to the anisotropy axis/velocity parallel to the anisotropy axis), so a magnitude of 1.0 means no anisotropy.

```
@AnisotDir
  0.000      0.000      0.000
@
  -7.000      9.000
  -5.000     11.000
   8.000     34.000
  9   9   14
-7.000     -5.000      8.000     11.000     0.00     1.000
-5.000     -5.000      8.000     11.000     0.00     1.000
```



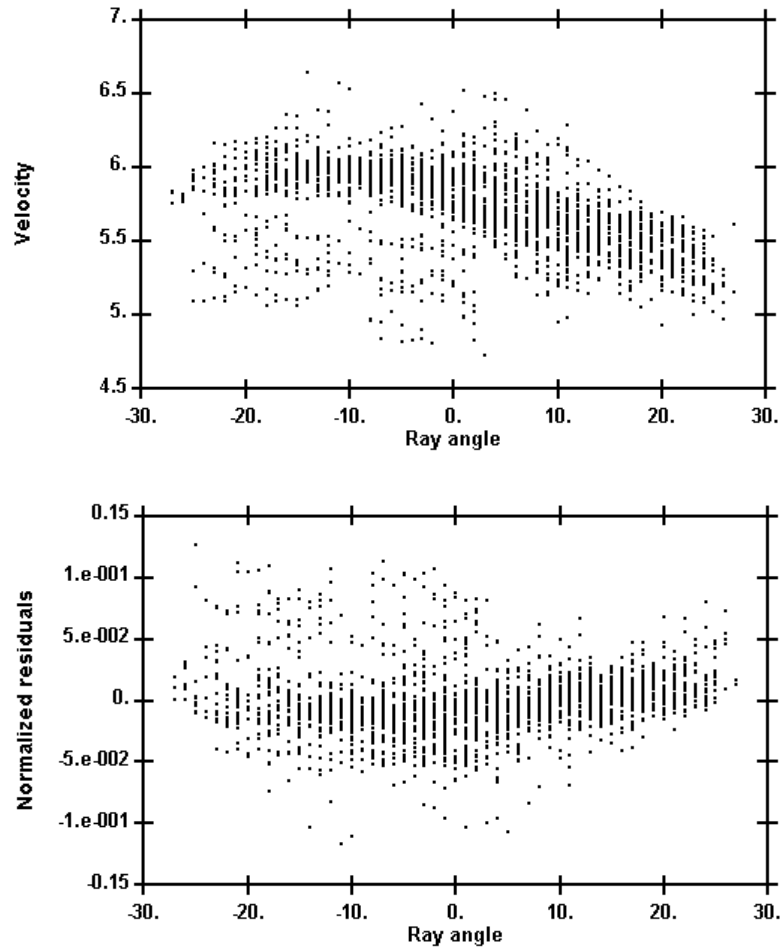


## Example 2. Ambiguous evidence of anisotropy, unconfirmed by residuals

The plot of velocity versus ray angle shown to the right indicates some dependence of velocity on ray angle, with a maximum velocity at  $-18^\circ$ .

However, the plot below it does not indicate a strong dependence of normalized residual on ray angle.

Applying anisotropy did not significantly improve the RMS residual, so the final inversion was performed without anisotropy. With the maximum velocity at  $-18^\circ$ , the positive Z axis up, and the primary source-to-receiver direction in the direction of the positive X axis, the corresponding axis of anisotropy should be tilted from the positive Z axis by  $18^\circ$  in the positive X direction. Then the X,Y,Z components of the axis of anisotropy are  $\sin(18^\circ)$ ,  $\sin(0^\circ)$ ,  $\cos(18^\circ) = 0.310, 0.000, 0.950$ . The RMS residual was not improved by applying anisotropy. The RMS residual was 0.227 without anisotropy and 0.236 with anisotropy in the indicated direction. The RMS residual was 0.224 with anisotropy axis tilted in the opposite direction, with X, Y, Z components  $-0.310, 0.000, 0.950$ . These changes in RMS residual were not significant. The fractional change in RMS residual calculated from  $(0.236-0.227)/\text{ave}(0.236 \text{ and } 0.227)$  was 0.009, or about 1 pct.



The dependence of velocity on ray angle was caused by two factors other than anisotropy. One factor was that the receiver borehole was deeper than the source borehole, so more of the deeper raypaths where velocity was higher were sloping downward. The second factor, probably more important, was a high-velocity structure that sloped downward from the source to the receiver borehole.

When examining the effect of anisotropy of magnitude 1.08 with the maximum velocity plane at  $-18^\circ$  and the positive Z axis up, the velocity model rows related to anisotropy were as follows:

```
@AnisotDir
  0.310      0.000      0.950
```

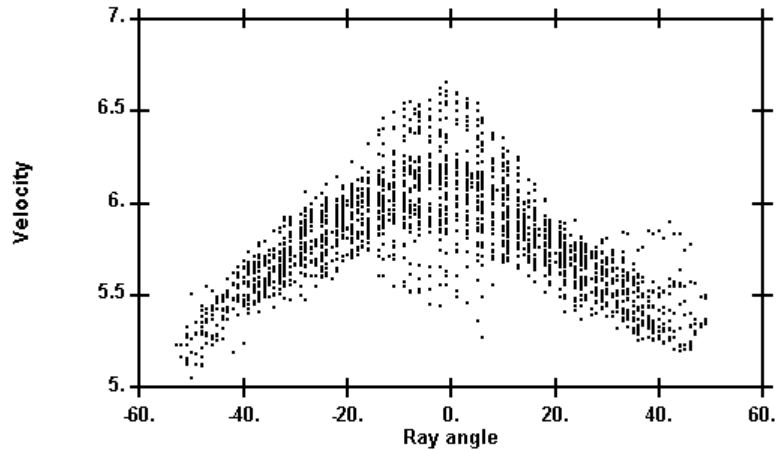
```

@
  -1.000    46.000
  -1.000     1.000
  12.000    40.000
  95    2    57
-1.000    -1.000    12.000    4.000    0.00    1.080
-0.500    -1.000    12.000    4.000    0.00    1.080

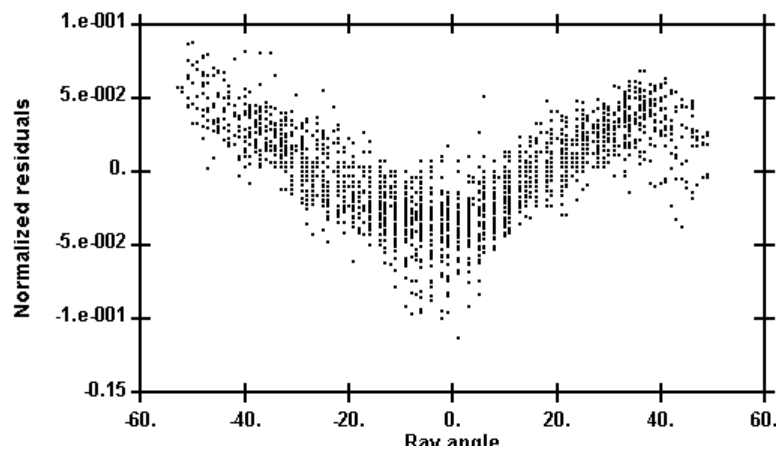
```

### Example 3. Anisotropy with maximum velocity in the horizontal plane

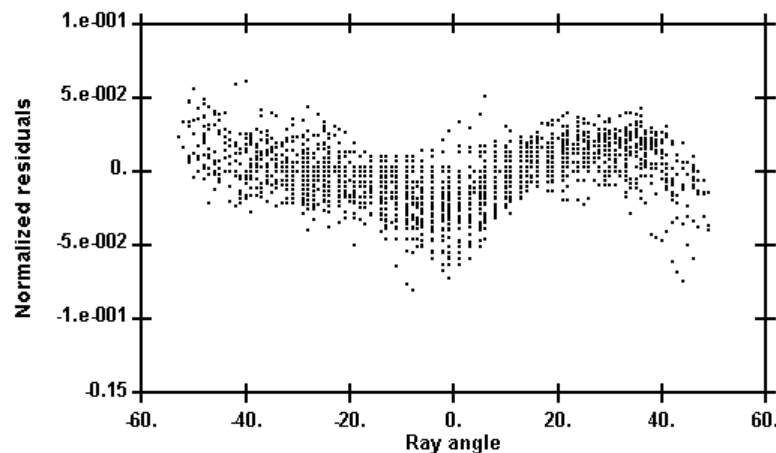
The plot to the right indicates that straight-line velocity depended on ray angle.



As confirmation, the plot below it shows that normalized residual depended on ray angle when the inversion was performed with no anisotropy.



When elliptical anisotropy with magnitude =1.2 was applied with the anisotropy axis parallel to the Z axis, the dependence of normalized residual on ray angle was reduced, as shown to the right



The RMS residual was 0.113 ms without anisotropy and decreased to 0.064 ms with anisotropy. The change in RMS residual calculated from  $(0.113 - 0.064) / \text{ave}(0.113 \text{ and } 0.064) = 0.55$  or 55 pct, a significant improvement.

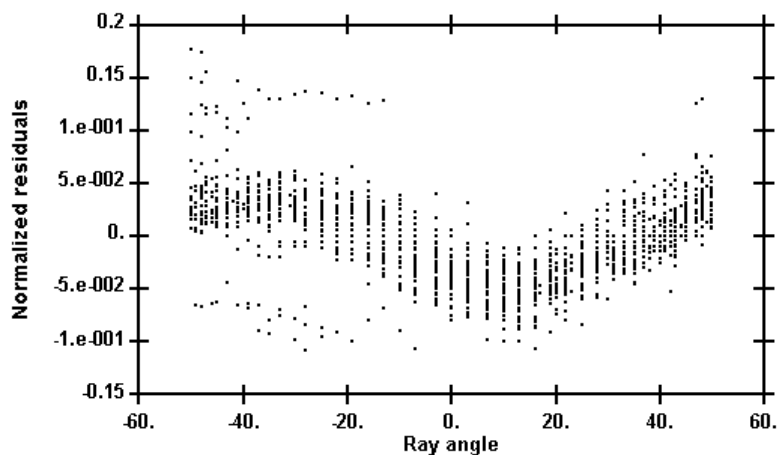
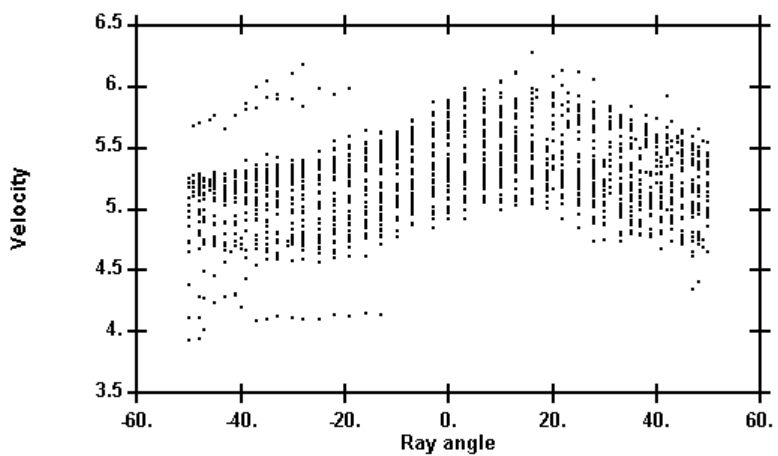
The rows in the velocity model file that controlled anisotropy were as follows:

```
@AnisotDir
  0.000      0.000      1.000
@
  0.000      2.000
  0.000      17.000
  13.000     37.000
  2   18   49
0.000      0.000      13.000      4.000      0.00      1.200
2.000      0.000      13.000      4.000      0.00      1.200
```

#### Example 4. Anisotropy with maximum velocity in a dipping plane

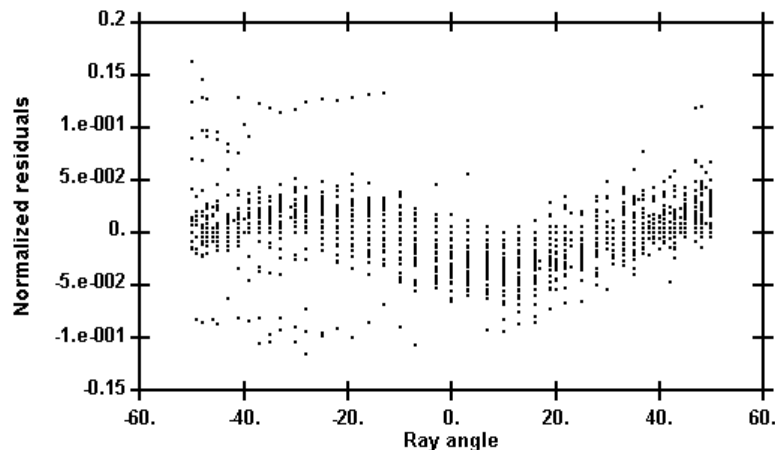
The plot to the right shows a dependence of velocity on ray angle with the maximum velocity in a plane at  $+18^\circ$  from horizontal. The positive angle means that the plane of maximum velocity rises from the source to the receiver borehole. The plane of maximum velocity is such that the Z coordinate increases when going from the source to the receiver if the positive Z axis is up (e.g. elevation) or decreases if the positive Z axis is down (e.g. depth).

As confirmation, the plot of normalized residual with no applied anisotropy versus ray angle is shown below it.



In this data set, the positive Z axis was up and the primary source-to-receiver direction was in the direction of the positive X axis. When elliptical anisotropy of strength 1.08 with anisotropy axis tilted  $18^\circ$  to the left of the positive Z axis was applied, the dependence of normalized residual versus ray angle decreased, as shown in the plot below.

The RMS residual with no anisotropy was 0.080 ms. It was 0.065 ms with anisotropy. The fractional decrease in RMS residual calculated from  $(0.080 - 0.065)/\text{ave}(0.080 \text{ and } 0.065) = 0.21$ , or 21 pct. This decrease is significant. In general, improvements greater than 10 pct are significant, in the author's experience.



The rows in the velocity model file to implement this anisotropy are shown below. The X component of the anisotropy axis is -0.31 because the anisotropy axis must tilt to the left of the positive Z axis by  $18^\circ$  to make the anisotropy plane Z increase from source to receiver, and  $\sin(-18^\circ) = -0.31$ . The Z component of the anisotropy axis is  $\cos(-18^\circ) = 0.95$ . The positive Z-axis points upward, so this slope is upward from source to receiver.

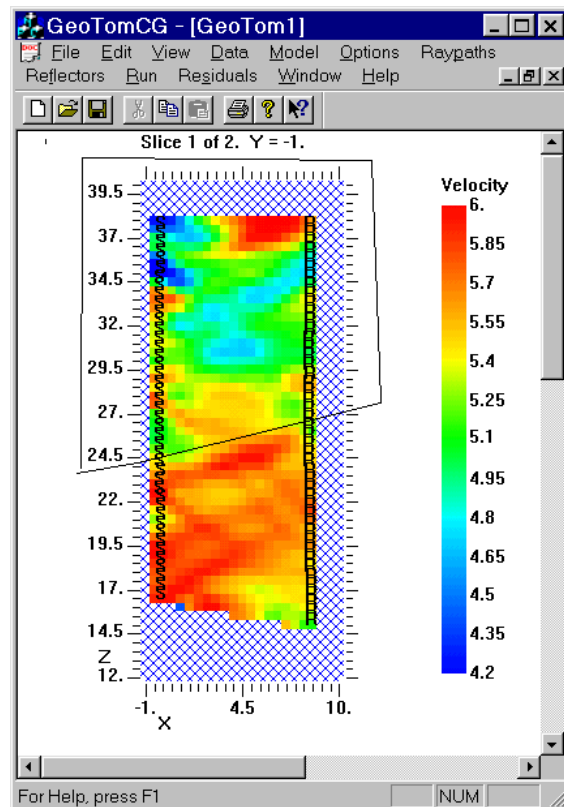
```
@AnisotDir
  -0.310      0.000      0.950
@
  -1.000     10.000
  -1.000      1.000
  12.000     40.000
  23  2  57
-1.000     -1.000     12.000     4.000     0.00     1.080
-0.500     -1.000     12.000     4.000     0.00     1.080
```

### Example 5. Nonuniform anisotropy and editing the velocity model accordingly

This example will consider the common case of anisotropy absent near the surface in soft material but present in deeper hard rock. One method to prepare a velocity model file is to first generate a file with uniform anisotropy of the magnitude desired for the deeper part, then use the **Model, Edit** option to change the anisotropy value near the surface to 1.0. The **Model, Edit** option allows the user to draw a dividing line between the shallower and deeper sections. The dividing line can be sloping.

The dialog box below shows the Edit model options set to make the anisotropy 1.0 in the region bounded by the editing line that will be drawn by the user. The **Set grid point anisotropies** option is set to **Through** and the value is set to 1. Then click **Do more editing before saving**.

Next draw a bounding line (see section "Editing the Model") around the upper region where the anisotropy = 1.0. An example is shown below. Note that the bounding line can be slanted and you can draw outside of the colored area. When completing the bounding line, right click to close the boundary and return to the **Edit model** dialog box. Then click **Save edited model this time** to save the edited model. You will be asked to name the new velocity model.



## COMPARING VELOCITY MODELS

GeoTomCG offers options for comparing velocity models by calculating differences or ratios. First open a velocity model in the usual way with **Model, Open**. That model is called Model1 in the comparison. Next click on **Model, Compare**. A dialog box will open. Select one of the following:

**Difference of Model1 and Model2,**  
**Ratio of Model1/Model2, or**  
**Ratio of Model2/Model1.**

Model2 is the model that will be read as the second model in the comparison, and Model3 is the model that will be calculated and displayed or saved. You can also enter multipliers for the models. You can find the average of Model1 and Model2 by taking  $0.5 \times \text{Model1} + 0.5 \times \text{Model2}$ . You can convert a velocity of m/ms to m/s by taking  $1000 \times \text{Model1} + 0 \times \text{Model2}$ .

After selecting the type of comparison and the multipliers, then click on either **Calculate and Save Model3** or **Calculate and Display Model3**. Enter a file name for Model2 that will be read. The number of grid-points in all directions must be consistent for Model1 and Model2. If you selected **Calculate and Display Model3**, Model3 will be displayed in the usual way. If you selected **Calculate and Save Model3**, you will be asked for the name of the Model3 file that will be written. In either case, Model3 replaces Model1 in memory. The velocity models can be large, so having two velocity models in memory at the same time is not desirable.

## READING A SERIES OF DATA FILES AUTOMATICALLY

A series of data files in the 3dd or 3dw format can be read and processed automatically, without the user providing additional input once the procedure has been started. First read one data file and one initial model, and select the inversion parameters such as the number of straight and bending-ray iterations. To save raypath files, click **Raypaths, Save raypaths** and enter any file name. Then click **Data, Automate data reading**, check the box for **Automate data reading**, and click **OK**. Clicking **Data, Open data file** will open a dialog box for selecting multiple data files. You can select up to 25 files with up to 2048 total characters, but no file path should exceed 100 characters. Use **Ctrl** and **Shift** keys in the usual manner for selecting multiple files in Windows. Click **Open**. The program will read a selected data file and the model file selected previously. The model file is reread each inversion to avoid storing both the initial and calculated models. The program performs an inversion without further user input. The calculated model, residual, and raypath files are given the name of the data file except the extension is 3mq, rsd, and ray, respectively.

After the first inversion, the program follows the same procedure with each of the selected data files. The order for processing data files might not be the same as the order they were selected. Be sure all the inversion parameters are set correctly because the program does not take any input once it starts. After all inversions are completed, the program displays a message box with the message "Program execution completed."

## **SPECIAL PROCEDURES FOR ATTENUATION TOMOGRAPHY**

(adapted from pages 43-47 of Bureau of Mines RI 9617)

Attenuation tomography requires inversion of measured amplitudes to compute the attenuation distribution. It is more complicated than travelt ime inversion because data reduction must be carried out before amplitude data can be processed by GeoTomCG and there are additional complexities in interpreting results.

Amplitude computations are based on the equation

$$-\ln(A/A_0) = \int_S^R \alpha dl \quad (1)$$

which states that the logarithmic amplitude decrease (or total attenuation) from source S to receiver R is the line integral of the attenuation rate  $\alpha$  along the path from S to R. A and  $A_0$  refer to the amplitudes at the receiver and near the source, respectively. The equation is analogous to that for calculating travelt ime on the basis of slowness (1/velocity). This analogy allows the program to calculate attenuation coefficients in the same way it calculates slowness, after the amplitude data have undergone proper data reduction.

Because GeoTomCG is most commonly used for travelt ime rather than amplitude inversion, it has certain features that are convenient for velocity imaging, but not necessarily for attenuation imaging. For example, although the calculations involve slowness, GeoTomCG displays its inverse, namely velocity. In the same way, attenuation calculations involve the attenuation rate, but GeoTomCG displays its inverse, namely skin depth or attenuation distance.

Model files created by the program contain attenuation distance, with units determined by those of the data. For example, with locations specified in meters, and reduced amplitudes in decibels, the model values will have units of m/dB. Physically this refers to the distance a wave travels before being attenuated by 1 dB (i.e., before its amplitude is decreased to  $10^{-1/20}$  (roughly 90%) of its initial amplitude.)

### **Ray Tracing in Amplitude Inversions**

Curved rays for amplitude inversion should not be generated by amplitude inversion because raypath curvature is controlled by the velocity distribution, not the attenuation distribution.

### **Interpretation of Attenuation Tomograms**

The usual pitfalls related to nonuniqueness apply equally to velocity and attenuation imaging. Here we will focus on considerations specific to attenuation images.

Perhaps the most important consideration is the inability to distinguish between scattering effects and intrinsic attenuation, which can have surprising consequences. For example, consider an air-filled subsurface void, such as a tunnel. For EM waves, the intrinsic attenuation in air is

much less than that in rock, and one might expect larger amplitudes for rays passing through the void. However, the strong contrast in physical properties across the void boundaries results in very strong scattering and a sharp decrease in the amplitude of the transmitted wave. Depending on various factors including the size of the void, the balance between scattering losses and lower attenuation in air may result in positive or negative amplitude anomalies, or even no anomaly. In addition, interference patterns may arise from scattering off different parts of an anomaly, resulting in amplitude maxima and minima that are not correctly computed by the ray-based methods of the program.

An additional potential problem relates to curved-ray processing. In a medium with sufficiently strong variation in physical properties, wave energy can be focused just as light through a lens, resulting in strong amplitude anomalies. Such anomalies will be incorrectly translated by the program into regions with anomalously high or low intrinsic attenuation in a tomographic reconstruction unless curved raypaths from a velocity inversion are used for the attenuation inversion.

## **What Is Amplitude?**

### Elastic Wave Amplitudes

The elastic wave equation, derived from stress-strain relations for elastic solids, describes how displacements propagate in space and time. Consider a point in the rock mass with position specified by coordinates  $(X, Y, Z)$ . As an elastic wave passes, the rock is locally compressed or distorted, and as a result the point at time  $t$  may have moved to location  $(X+dX, Y+dY, Z+dZ)$ . The displacement vector at time  $t$  is then  $(dX, dY, dZ)$ . An instrument that detects displacements in the  $X$  direction, for example, will measure a value  $dX$  at time  $t$ . Continuous recording produces a time series  $dX(t)$ . The total displacement at any time has a magnitude  $(dX^2 + dY^2 + dZ^2)^{1/2}$ . Determining this displacement requires three perpendicular single-axis sensors or a triaxial sensor.

Many types of sensors are designed to measure not displacement, but one of its time derivatives, velocity or acceleration. Engineering geophones typically produce a voltage proportional to ground velocity. (Ground velocity should not be confused with wave velocity; the former is the speed with which the ground moves at the measurement point, and the latter is the speed with which the disturbance propagates through the rock). Other sensors, for example hydrophones, use piezoelectric materials to produce a voltage proportional to pressure. Combining data from different types of sensors for amplitude inversion requires careful cross-calibration and is generally not recommended.



## Electromagnetic Wave Amplitudes

The EM wave equation, derived from Maxwell's equations, describes how components of the electric field  $E$  and magnetic induction  $B$  vary in space and time (Nabighian, 1988, Ch. 4). Sensors are typically electric dipole antennas in which a current is generated proportional to the parallel component of  $E$ , or magnetic dipole (loop) antennas in which a current is induced, proportional to the perpendicular component of  $\text{dB}/\text{dt}$ . Measurement of the total field  $E$  or  $\text{dB}/\text{dt}$  requires three orthogonal sensors.

## Radiation

Wave behavior for both seismic and EM sources is rather complex in the immediate vicinity of the source. The wavelength of the energy is often used to distinguish between "near-field" (much less than one wavelength from the source), "far-field" (many wavelengths distant from the source), and intermediate zones. For EM waves, the far-field and intermediate zones are often referred to as the radiation zone and the induction zone, respectively. For geophysical tomography, measurements are most often made in the far-field, and the calculations of GeoTomCG assume this to be the case.

Wave amplitudes may vary with direction for certain types of sources. For electric dipoles (straight antennas) the magnitude of the  $E$  and  $B$  fields produced in the far-field is proportional to the sine of the angle between the dipole and the propagation direction (i.e., amplitude is maximum perpendicular to the antenna and minimum parallel to it.) Similarly, for magnetic dipoles (loop antennas, with the magnetic dipole perpendicular to the loop), amplitude is proportional to the sine of the angle with respect to the dipole. Seismic sources often produce amplitudes nearly independent of direction but the amplitudes may have some other dependence. Explosive sources tend to produce amplitudes independent of direction. Continuous or nearly continuous sources such as continuous coal miners, tunnel boring equipment, or drilling rigs can be used for collecting amplitude data, but it is not always obvious what the radiation pattern of such sources may be, and care is required.

## **What Is Attenuation?**

The  $E$  and  $B$  fields of an EM wave, and the displacements of an elastic wave, decrease in magnitude as the wave travels. This decrease, or attenuation, results from three principal causes: geometric spreading, absorption, and scattering.

## Geometric spreading

Waves emanating from a point source expand spherically (in a uniform medium) with time. Neglecting for now absorption and scattering, the total energy of a spherical wavefront remains constant as the wave expands, and the energy per unit area diminishes accordingly, following the well-known inverse-square law. Energy is generally proportional to the square of amplitude (displacement,  $E$ ,  $B$ , or their time derivatives); consequently amplitude decays as  $1/r$  (where  $r$  is the distance from the point source).

For an infinitely long line source, waves expand cylindrically rather than spherically. In this situation, energy (per unit area on the wave front) decreases as  $1/r$  and amplitudes as  $r^{-1/2}$ . This holds approximately for line sources of finite length, if  $r$  is much less than the source length. Conversely, if  $r$  is much greater than the source length, the source “looks like” a point dipole from the receiver and amplitudes diminish approximately as  $1/r$ . In a layered medium, waves may be “trapped” in a particular layer (a “waveguide”), and cylindrical spreading may be a good approximation.

### Absorption (intrinsic attenuation)

In an electrically-conductive material, EM waves induce eddy currents, and consequently some of the wave energy is dissipated or converted into heat. In an imperfectly elastic material, seismic waves lose some energy to internal friction. Both of these types of attenuation are determined by the physical properties of the medium (in contrast to geometric spreading, which is determined principally by experimental geometry). This intrinsic attenuation is of chief interest for geophysical imaging. In a uniform attenuating medium, intrinsic attenuation causes an exponential decay of amplitude with distance, i.e., amplitude is proportional to  $\exp(-\alpha r)$ , where  $\alpha$  is the attenuation coefficient.  $\alpha$  has dimensions of inverse length, and units are typically nepers/m (Np/m) or decibels/m (dB/m). The reciprocal of  $\alpha$  is called the skin depth or the attenuation distance  $\delta$ , representing the distance over which amplitude decreases to a fixed fraction of its initial value.

### Scattering

In a medium with discontinuities (e.g., layering or other sharp lithologic boundaries) wave energy is partitioned at each interface into reflected and transmitted fractions. For elastic waves, this partition may involve mode conversion. For example a compressional wave incident on a boundary may give rise to compressional and shear waves, both transmitted and reflected. The technique of diffraction tomography attempts to utilize scattered wave energy to image discontinuities (Devaney, 1984). In GeoTomCG, energy loss due to scattering is not distinguished from intrinsic attenuation.

## **Quantifying Amplitudes**

Far-field wave amplitudes can be measured for either pulsed or continuous energy sources and used for tomographic inversion.

Pulsed systems produce wave energy in short bursts; examples include hammers, explosives, air guns, sparkers, and pulsed radar systems. Measurements are made in the time-domain; that is, ground motion or EM fields are measured as a function of time, and the arrival of a wave front is recorded as one or more peaks and troughs in the time-series. Amplitude is usually characterized in terms of peak height, trough depth, or the difference of these. Recorded amplitudes generally depend on factors such as detector sensitivity and amplification, and these calibration factors must be known to determine actual ground motion.

Continuous systems generate continuous or quasi-continuous wave trains; examples include continuous-wave radio transmitters, drilling rigs, and mining machines. Measurements are made in the frequency domain; that is, amplitude is determined as a function of frequency. Alternatively, measurements may be made in the time domain and transformed to the frequency domain. Amplitudes are then characterized in terms of peaks in the amplitude spectrum, or by integration over all or part of the spectrum. Again the amplitudes thus determined depend in part on the characteristics of the measurement and recording system.

Because of the exponential nature of intrinsic attenuation, it is often convenient to characterize amplitudes on a logarithmic scale rather than a linear one. In fact, as described later, the program requires amplitude data to be scaled logarithmically. Logarithmic scaling naturally involves dimensionless ratios, or relative quantities, rather than absolute quantities; amplitudes are thus normalized to some quantity, and then the logarithm is taken of the ratio. Amplitude ratios are most often expressed in decibels (dB), defined as 20 times the decimal logarithm of the amplitude ratio. Thus 20 dB corresponds to an amplitude ratio of 10. Alternative units are nepers, defined as the natural logarithm of the amplitude ratio ( $1 \text{ dB} = 0.1151 \text{ Np}$ ).

Some measurement and recording systems report amplitudes in dB (relative to some reference amplitude.) For example, a continuous-wave (frequency-domain) radio receiver using electric dipole antennas may display amplitudes in dB relative to one millivolt. If the potential difference due to the electric field at the receiving antenna is 10 millivolts, the value displayed will be  $20\log_{10}(10/1)$ , or +20 dB. If the receiver displays a value of -10 dB, the field is  $10^{(-10/20)}$ , or 0.316 mV.

A common practice is to place one or more receivers close to the source, and to normalize all other amplitudes at greater distances to this “source amplitude”  $A_0$ . If the linear amplitude measured by a particular receiver is  $A$ , the ratio  $A/A_0$  corresponds to the amplitude reduction along the propagation path, which is the total attenuation due to spreading, absorption, and scattering. This attenuation can be expressed in decibels as  $dA=20\log_{10}(A/A_0)=20\log_{10}(A)-20\log_{10}(A_0)$ . A value of zero indicates no attenuation ( $A=A_0$ ); greater negative values correspond to greater total attenuation. For a plane wave (with no spreading) the total attenuation is equal to the intrinsic attenuation rate (dB/m) times the distance traveled. This is analogous to the travelttime equation: total travelttime is equal to the wave slowness ( $1/\text{velocity}$ ) times distance.

## Amplitude Data Reduction

The program performs its calculations in the same way for both amplitude data and traveltimes data. Therefore it is necessary to reduce the amplitude data to a standard form appropriate for the calculations. For traveltimes, the program calculates the integral of wave slowness along each raypath. If, instead of slowness, we integrate the attenuation coefficient  $\alpha$  along each raypath, the result is a calculated logarithmic amplitude drop (or total attenuation)  $dA$ . More specifically, it is the logarithmic amplitude drop  $dA_A$  due to intrinsic attenuation, or absorption (as opposed to geometric spreading and scattering.) This, then, is what we must provide as data for the program. In order to calculate  $dA_A$ , we must correct  $dA$  for the loss due to geometric spreading  $dA_G$ :  $dA_A = dA - dA_G$ . Because we cannot correct for scattering loss  $dA_S$ , any losses due to scattering will show up in the tomographic image as absorption. The following is a step-by-step description of the corrections that may be required.

1. Determine “raw” amplitudes. The time-picking program TomTime provides an option for recording the unprocessed amplitudes for one or three peaks following the first arrival. If you recorded three, you may use whatever combination of these peaks you choose as the raw amplitude. For example, you can use the amplitude of the first peak only, or you can use the combination of the first and second peaks to determine the peak-to-trough amplitude. For time-domain records with pulse sources, determine the peak height, trough depth, or combination of these for the arrival of interest. Avoid using reflected arrivals unless you know the position of the reflector, because the program will not be able to determine the correct raypath.

For continuous-source frequency-domain data, different wave modes cannot be identified directly by their separation in time, but you may be able to identify particular frequency ranges to use for amplitude determination. For example, if you are using a drill bit or mining machine as a source, you may be able to isolate part of the spectrum where the desired signal is strong and other mechanical and electrical noise sources are minimized. However, it is unlikely that you will be able to separate direct from refracted or reflected energy. Determine the peak amplitude for the selected frequency range, or integrate over the frequency range to quantify the amplitude, for each source-receiver combination.

For either time or frequency domain determinations, you will probably first obtain a raw measure of amplitude expressed in volts, reflecting the receiver sensitivity, amplification, and wave amplitude. If different system parameters were used for different raypaths, you will need to calibrate the data.

2. Calibrate amplitudes. Seismic receivers generally produce a voltage proportional to displacement, velocity, or acceleration, and the constant of proportionality is termed the sensitivity. For example an accelerometer may have a sensitivity of one volt per  $g$ , where  $g$  is the average gravitational acceleration at the Earth’s surface,  $9.8 \text{ m/s}^2$ . It is not unusual with multichannel recording systems to use receivers with different sensitivities simultaneously, for example at different distances from the source. In such a situation, you must divide each raw amplitude by the corresponding sensitivity. Similarly, you may have different gain settings or amplification for different raypaths and you must correct for this. Gain is often expressed

logarithmically, in terms of dB. A gain of 20 dB, for example, indicates that measured values have been multiplied by a factor of ten.

3. Make spreading correction. Ordinarily it is safe to treat sources as points if the source-receiver distance is much greater than the source dimensions, and to treat spreading as spherical. If, however, the point-source approximation is not valid, or if wave energy is confined to a waveguide, a cylindrical spreading correction may be more appropriate.

For spherical spreading, amplitudes decay as  $1/r$ ; for cylindrical spreading the rate is  $r^{-1/2}$ . To compensate for this loss, multiply each linear amplitude by the corresponding source-receiver distance (or by the square root of the distance for cylindrical spreading). (This multiplication adds a factor of length to the units of amplitude, but this will be normalized out later.) The result is an estimate of what the amplitude would be for a plane (non-spreading) wave for each ray path.

If your data are logarithmic (dB or Np), convert the appropriate distance term ( $r$  or  $r^{1/2}$ ) to the same log units (e.g.,  $20\log_{10}(r)$ ), and add to the measured amplitude for each ray path.

4. Make radiation correction. If your energy source has a radiation pattern that is not monopolar (i.e., if it generates waves with amplitudes that vary according to direction of propagation), you need to correct for this. For electric or magnetic dipole antennas, for example, the radiation is often proportional to  $\sin(\theta)$ , where  $\theta$  is the angle between the dipole axis and the line from the source to the receiver. (Tuned antennas, such as a half-wavelength antenna, have a stronger dependence on angle.) Shear waves generated by a vertically moving borehole hammer have a maximum amplitude perpendicular to the hammer motion with amplitudes proportional to  $\sin(\theta)$ , where  $\theta$  is the angle between the hammer motion and a line from the source to the receiver. Compressional waves from borehole explosive or sparker types of sources can be expected to have amplitudes nearly independent of direction for raypaths that are not close to the direction of the borehole. For other types of sources, such as mining machinery, it may not be apparent what sort of radiation to expect, and it may be necessary to determine it by experiment. If amplitude dependence on angle is  $F(\theta)$ , it must be compensated for by calculating  $\theta$  for each source-receiver combination, and dividing the corresponding amplitude by  $F(\theta)$ . This will increase the amplitude for unfavorably oriented rays, and not change amplitudes for favorably oriented rays, so that the corrected values are close to what would have been determined for a monopolar source.

With logarithmic amplitude data, convert the angular correction factor to the same log units (dB or Np). Because sine and cosine are always less than or equal to 1, their logarithms will be negative. Subtract this negative number from its corresponding amplitude, in effect adding a positive correction.

5. Make receiver sensitivity correction. Often receivers measure only a single (Cartesian) component of the amplitude; for example many geophones measure only the vertical component of ground motion, and an electric dipole antenna measures only the component of electric field parallel to the antenna. The full amplitude can be determined from triaxial geophones or from hydrophones for compressional waves, but it can be approximated for crosshole single-axis data

by assuming that the propagation path is a straight line from the source to the receiver. Further assuming the nature of the wave is known, e.g., compressional or shear, we can determine the expected orientation of particle displacement, velocity or acceleration, or E and B fields. Calculate the angle  $\theta$  between this orientation and that of the receiver axis for each ray path, and divide the measured amplitude by  $\cos(\theta)$ . This will provide an estimate of the amplitude that would have been measured by an optimally-oriented single-axis receiver at the same location.

If you have logarithmic rather than linear amplitudes, handle the correction as described above for source radiation.

6. Compute logarithmic relative amplitude. If your corrected data are linear amplitudes  $A$  (e.g., displacements, velocities, voltages, etc.), and you have measured (and corrected) the source amplitude  $A_0$  using a receiver placed near the source, calculate the amplitude ratio  $A/A_0$ , and convert to either nepers (by taking the natural log) or dB (by taking the decimal log and multiplying by 20). The result should be a negative number in all cases.

If you have linear amplitude data without a direct estimate of  $A_0$ , you can estimate it indirectly. Begin by converting the corrected linear amplitudes to Np or dB. Plot them as a function of source-receiver distance; generally you will see something like a linear trend with a negative slope. Extrapolating this trend backwards to a position near the source will give you an estimate of  $A_0$  (in the same Np or dB units). Subtract this value from all of your logarithmic amplitudes; the result should be a negative number in all cases.

If you have logarithmic amplitude data to begin with (i.e., if your systems records amplitudes in dB relative to some arbitrary reference value), and you have a measured value for  $A_0$ , subtract this from all other data to obtain the logarithmic amplitude ratios. If your data are logarithmic and you do not have a measurement for  $A_0$ , determine it indirectly by extrapolation as described above. Again the result should be a negative number for all rays.

7. Change sign. The logarithmic relative amplitudes obtained by the steps above will all be negative, reflecting the fact that measured amplitudes are all less than the source amplitude, due to attenuation. To put the data into a form analogous to travelttime data, simply change the sign to positive; the values now indicate the total attenuation (or amplitude drop) along each ray path. GeoTomCG will calculate corresponding values as line integrals of attenuation rate (dB/m or Np/m) along the ray paths.

## ILLUSTRATED EXAMPLES OF AMPLITUDE PROCESSING

Three examples will be given. The first is the simplest, and is appropriate for many seismic systems where the amplitude is expressed directly as a magnitude, not in dB as the ratio of magnitudes. The second is appropriate for many GPR systems where the amplitude is expressed as a magnitude, not in dB. The third is for GPR systems in which the amplitudes are measured in dB relative to some standard magnitude.

All of these example time and amplitude (\*.tam) files and the Excel worksheets (AmpExamp.xls) in which they were generated are provided as part of the sample files.

### Example 1, Spherical Source Emission

The first example begins with the unprocessed amplitudes in the file SeisUnpr.tam. It contains the demo1 time data, plus synthetic amplitude data. The amplitude data were generated by assuming that  $A_0 = 10000$  and that the amplitude is proportional to  $(1/L)\exp(-0.05*L)$ , where  $L$  is the distance along the raypath. The  $1/L$  dependence is appropriate for sources having a spherical wave output. The coefficient of attenuation is  $0.05$  Np/ft. The units are nepers (Np) rather than dB because the coefficient is in the expression  $\exp(-0.05*L)$ , and the amplitudes are expressed in values rather than in dB ratios. Nepers are ratios, like dB, but are the natural logarithm of amplitude ratios, and so can be calculated more directly from exponentially decreasing amplitudes. The source and receiver coordinates are in feet for this data set. The absence of any angular dependence is appropriate for many seismic crosshole P-wave sources.

In the following example, pretend that  $A_0$  and the coefficient of attenuation are unknown, and must be calculated from the data. Calculating them requires the following tasks:

1. Decide the geometrical spreading factor. It is  $1/L$  for point sources and  $1/\sqrt{L}$  for cylindrical sources. For this example, use  $1/L$ . The  $1/L$  spreading factor is much more common than  $1/\sqrt{L}$  because a cylindrical source that is small compared to the source-receiver separation will act as a point source.
2. Decide if there is an angular dependence to the source emission pattern. For this example, assume there is none.
3. Decide if there is an angular dependence to the receiver sensitivity. For this example, assume there is none.

Next consider the equation to solve for  $A_0$  and  $\alpha$ . For the example, it is

$$A = (A_0/L) \exp(-\alpha L),$$

where  $A$  is the measured amplitude, and  $\alpha$  is an average attenuation coefficient that can be used to determine the initial estimate. The inverse of attenuation coefficient is used as the input to GeoTomCG.

Some spreadsheets allow the user to enter the equation and solve for  $A_0$  and  $\alpha$  directly, using the columns of distances and measured amplitudes. Assume for this example that we can do only linear regression. Then the equation should have the form  $Y = X_0 + b X$ .

(For traveltimes,  $Y$  is the measured time,  $X_0$  is 0,  $b$  is  $1/\text{velocity}$ , and  $X$  is raypath length.)

To transform the equation into the desired form, it can be rewritten as

$$\ln(AL) = \ln(A_0) - \alpha L,$$

which has the desired form with  $Y = \ln(AL)$  and  $b = -\alpha$ .

We calculate  $\ln(AL)$  for each raypath, then solve for  $\ln(A_0)$  and  $\alpha$  by performing a linear regression on  $\ln(AL)$  versus  $L$ . In the example, we get  $b = -0.05$ , so  $\alpha = 0.05$ , and  $\ln(A_0) = 9.21034$ , so  $A_0 = 10000$ .

To make the equation analogous to the traveltime equation, we need to transform it into the form  $Y = bL$ , so we write it as

$$\ln(AL) - \ln(A_0) = -\alpha L.$$

Because  $\alpha$  is positive, we rewrite the equation as

$$\ln(A_0) - \ln(AL) = \alpha L.$$

Then the processed amplitudes that are analogous to traveltimes are  $\ln(A_0) - \ln(AL)$ . This expression is correct if we want the units for the attenuation coefficient to be  $\text{Np}/\text{length}$  (or  $1/\text{length}$ ) rather than  $\text{dB}/\text{length}$ .

Note that  $\text{time} = (1/\text{velocity}) * \text{length}$ , and the calculated parameter is velocity. Thus, we need to input as the initial guess  $(1/\alpha)$  or 20, and we will calculate the reciprocal of the attenuation coefficient.

The data file containing these processed amplitudes is `SeisProc.tam`. When `GeoTomCG` is run with these amplitudes as data, straight-ray iterations quickly find that the inverse of the attenuation coefficient is a uniform 20. The model file `Dem1NoAn.3dm` can be used as an initial model.

## Example 2, Nonspherical Source Emission

Now consider another example, with an additional complication often appropriate for GPR data. Assume that the emission from the source is proportional to the cosine of the angle between the raypath and a horizontal line. Similarly, assume that the sensitivity of the receiver is proportional to the cosine of the angle between the raypath and a horizontal line. These assumptions are often appropriate for crosshole GPR systems with vertical source and receivers. Assume that the average angle of the raypath (a straight line from source to receiver) is adequate for consideration of angular effects.



Then the equation describing the amplitude expressed in values such as millivolts (mV), rather than dB, is

$$A = (A_0/L)(\cos \phi)(\cos \phi)\exp(-\alpha L),$$

where  $\phi$  is the angle between the raypath and a horizontal line, assuming the source and receiver are both vertical. Here  $\alpha$  is still in units of Np/ft because this formula has only  $\alpha$  and  $L$  in  $\exp(-\alpha L)$ .

The sample data file GPRUnpNp.tam contains synthetic amplitude data generated with this formula. To calculate  $A_0$  and  $\alpha$  from these data with linear regression, we can write the equation as

$$\ln(AL) + \ln(1/\cos \phi)(1/\cos \phi) = \ln(A_0) - \alpha L,$$

which can be simplified to

$$\ln(AL) - 2\ln(\cos \phi) = \ln(A_0) - \alpha L.$$

Thus, we would calculate  $\ln(AL) - 2\ln(\cos \phi)$  for each raypath, then calculate an overall  $\ln(A_0)$  and  $\alpha$  from linear regression on  $\ln(AL) - 2\ln(\cos \phi)$  versus  $L$ . This regression yields  $\alpha = 0.05$ , and  $\ln(A_0) = 9.21034$ , so  $A_0 = 10000$ . As before, we could use  $1/\alpha = 20$  as the initial estimate for the attenuation inversion.

The processed amplitudes for GeoTomCG input are  $\ln(A_0) - \ln(AL) + 2\ln(\cos \phi)$ . These processed amplitudes are in the file GPRProNp.tam. GeoTomCG would calculate the distribution of the inverse of the attenuation coefficient, with the attenuation expressed in units of Np/length (or 1/length). Straight-ray iterations with GPRProNp.tam as data and any reasonable model such as Dem1NoAn.3dm yield  $1/\alpha = 20$ , with the perfect fit expected for these synthetic data.

### **Example 3, Nonspherical Source Emission, Amplitude in dB**

The third example is illustrated by the file GPRUnpdB.tam. It is like the file for the second example, GPRUnpNp.tam, except that the amplitudes have been converted to dB relative to 1. The conversion of each amplitude  $A$  to dB was performed by taking  $20 \log(A/1)$  where  $\log$  is the logarithm to base 10. The dB units are relative, but are usually taken relative to 1 unit, so the fact that dB units are ratios is not always apparent. The 1 will be omitted from the remaining equations. Assume that we want the units of the attenuation coefficient to be dB/length, in this case dB/ft.

The basic equation is the same as for example 2, but we will take  $20 \log$  of both sides to obtain dB units. The equation is, as in example 2,

$$A = (A_0/L)(\cos \phi)(\cos \phi)\exp(-\alpha L).$$

To prepare for taking 20 log of both sides, the equation can be written as

$$(A/A_0)(L)(1/\cos \phi)(1/\cos \phi) = \exp(-\alpha L).$$

Then

$$20 \log [(A/A_0)(L)(1/\cos \phi)(1/\cos \phi)] = 20 \log [\exp(-\alpha L)] = -\beta L,$$

where  $\beta$  is the attenuation coefficient expressed in dB/length.

Separating the terms on the left and moving  $A_0$  to the right to make the form similar to that in example 2 gives

$$20 \log(A) + 20 \log(L) - 40 \log(\cos \phi) = 20 \log(A_0) - \beta L.$$

Here  $20 \log(A)$  is separated from  $20 \log(L)$  because  $20 \log(A)$  is the amplitude in dB, and this example assumes that the amplitude is already expressed in dB. Then the equation is

$$\text{dB} + 20 \log(L) - 40 \log(\cos \phi) = 20 \log(A_0) - \beta L,$$

where dB is the amplitude expressed in dB.

We can calculate  $\text{dB} + 20 \log(L) - 40 \log(\cos \phi)$  and perform a regression analysis on it versus  $L$  to calculate an overall  $20 \log(A_0)$  and  $\beta$ . This regression yields  $\beta = 0.43429$  and  $20 \log(A_0) = 80$ , so  $A_0 = 10^{80/20} = 10000$ .

The processed amplitudes for GeoTomCG input are  $20 \log(A_0) - \text{dB} - 20 \log(L) + 40 \log(\cos \phi)$ . These processed amplitudes are in the file GPRProdB.tam. An inversion calculates the distribution of the inverse of the attenuation coefficient, with the attenuation expressed in units of dB/length. Straight-ray iterations with GPRProdB.tam as data and any reasonable model such as Dem1NoAn.3dm yield  $1/\beta = 2.3026$ , with the perfect fit expected for these synthetic data.

Recall that  $1 \text{ dB} = 0.1151 \text{ Np}$ . Then  $\beta = 0.43429 \text{ dB} \times (0.1151 \text{ Np}/1 \text{ dB}) = 0.05 \text{ Np}$ , which is consistent with the previous example.

## DEMO DATA SETS AND TUTORIAL

The GeoTomCG program package provides two data sets for practice, field data in the Demo1 set and synthetic data in the 3DCylSynth2Defects set.

The Demo1 data set includes cross-borehole high-frequency seismic travel times for four vertical boreholes in the Colorado School of Mines Edgar Mine facility. The rock mass at this site consists primarily of gneiss intruded by granite. Laboratory measurements of P-wave velocity on intact specimens averaged 4.66 m/ms (16.4 ft/ms). The data were processed in two dimensions for each borehole pair by Jessop et al. (1994), and a geological interpretation of the results is presented in that paper. The data were reprocessed in three dimensions by Jackson et al. (1995), and issues of resolution are discussed therein. The objective of the initial study was to locate zones of fractured rock prior to a simulated stope leaching experiment.

First examine the data. Open a Demo1 data file by clicking **Data**, then **Open data file**. A standard Windows file selection dialog box will appear. Select either Demo1.3dd or Demo1WithLabels.3dd and click **Open**. Demo1WithLabels.3dd includes borehole labels described previously. The data will be displayed in a scrollable window. The display is not a line by line copy of the data file. The program reads the variables, then writes them in the scrollable window so as to reveal any problems with reading the data. The coordinates are in feet and times in milliseconds, so the velocity units are ft/ms. Click **OK** to accept the data.

Click **Data, Plot**, and a dialog box with plotting options will appear in the upper left corner of the screen. For the horizontal axis variable, select **Source-rec distance**. For the vertical axis variable, select **Measured time or amp**. Click **OK** to display the plot. The resulting time-distance plot shows a general trend of increasing travel time with increasing distance, and no obvious outliers. Again click **Data, Plot, and** select **Source-rec distance** for the horizontal axis variable. This time, select **Str-line vel or 1/atten** for the vertical axis variable. Click **OK** to display the plot. The resulting velocity-distance plot reveals the range of the velocities calculated for the individual measurements assuming straight rays. The calculated velocities extend above and below the laboratory data. Reduced velocities may be due to fracturing. The higher velocities may be due in part to overburden stress or to errors in travel-time picking with noisy data. Eliminating erroneous data is important because they can produce artifacts.

Next generate a suitable initial velocity grid-point model. A model must be opened before GeoTomCG can perform an inversion. Demo1.3dm and Demo1.3mq provide voxels of 2 x 2 x 2 ft and uniform 11 ft/ms velocity, but you can generate another model for this exercise. Click **Model, Generate model**. A dialog box with information about the current model (the default model if no other model has been opened) and boxes for entering new model parameters appears. For this exercise, accept the default model by clicking **OK**. A **Save as** dialog box allows you to enter a name and click **Save**. Then open that model by clicking **Model, Open model file** and selecting it in the dialog box. Click **Open**. The model parameters will be displayed in a scrollable window. The display shows the parameters read by the program, rather than a line by line copy of the model file, to reveal any problems in reading them. Click **OK** to accept the model.

Next select options for running the inversion. The default of 1 straight-ray followed by 7 bending-ray (curved-ray) iterations is often suitable for typical data sets, but a rapid straight-ray inversion to examine the raypath density and residuals can be useful with large data sets. Click **Options, Inversion parameters**. Enter the number of straight and curved ray iterations, and the minimum and maximum allowed velocities. Either leave the # of straight-ray and curved-ray iterations at 1 and 7, respectively, or you may run first a rapid inversion with 5 straight-ray iterations. Leave the default minimum and maximum allowed velocities. Click **OK**.

To save raypaths, click **Raypaths, Save Raypaths** and provide a name for the raypath \*.ray file to be saved. A raypath file contains the 3D coordinates of raypaths that can be read and shown while displaying the model. Saving raypaths is optional.

To run the program, click **Run** on the main menu, then click **Run standard inversion**. Enter file names for the new model file and for the residuals file. The RMS residual for each iteration is listed as the iteration is completed. A message box indicates when the inversion is complete.

After running the inversion, open the newly calculated model by clicking **Model, Open model**, and select it in the **Open** dialog box. Click **Open**. When the model parameters are shown in the scrollable window, click **OK** to accept the model. Then select **Model, Display model**. The model display dialog box will appear in the upper left corner of the screen. Select the XY, XZ, or YZ slice plane with the radio buttons. Select the slice number with the up-down control, which also indicates the slice position in the plane. Examine the computed velocity model by selecting **Velocity at points** for **Display what?** Click **Show slice**. The slice will be shown in a scrollable window. The source and receiver positions from the open data file are shown as S and R, respectively, in 2D displays. If **All** is selected under **Show sources and receivers?**, then the positions are shown for all sources and receivers, not just the ones within the slice being viewed. A title at the top shows the slice number and position. The slice number can be changed with Page Up and Page Down keys. The **Rays per point** cutoff in the lower left of the dialog box can be reduced below the default because of large variations in raypath density.

The tomogram contains strong variations in velocity. These variations are confined to the areas with nonzero sampling density; unsampled regions remain at the uniform default initial velocity, and appear as featureless, uniform regions of initial velocity with a crosshatched pattern. A prominent resolved feature is the low-velocity zone extending subhorizontally at a depth of about 20 feet, and a Y coordinate of -3 feet, with velocities much lower than those measured for intact specimens in the laboratory. Drilling showed this region to be strongly fractured (Jessop et al., 1994).

Selecting the **Rays per grid point** option for **Display what?** creates a plot of the number of raypaths affecting the calculated velocity for each grid point. Viewing this plot in the XY plane shows a high raypath density along and between the boreholes.

You can perform more iterations after viewing a computed model by reopening the data file and using the saved model file from the previous run as the initial model for the new run.

When a data set is comprised of several interlocking crosshole surveys, as in this example, the best resolution is in the planes between boreholes. It is therefore of interest to view plots of the

velocities in these planes. For the Demo1 data set, two of these planes are approximately perpendicular to the coordinate axes, and these two can thus be viewed by selecting slices near  $X=0$  and  $Y=-3$ . The other surveyed planes are oriented diagonally across the grid. To view contour plots of these planes, you must first note the coordinates of the boreholes of interest by opening the data file Demo1.3dd. Examine the data in the scrollable window before clicking **OK**. The raypaths numbered from 98 to 217 were collected using the borehole pair with X, Y coordinates (-5.84, -2.4) and (0, 9.29), and Z coordinates between 10 and 20.

To generate a plot of this rectangular area, select **Model, Extract vertical slice**. It extracts a vertical slice over the height of the grid from a vertical line A to a second vertical line B. The user can accept the default grid corner to corner coordinates or enter the values for the X and Y coordinates for the two lines in a dialog box. For line A, enter X, Y = -5.84, -2.4. For line B, enter X, Y = 0.0, 9.29. Click **OK** and provide a name for the diagonal slice file. The diagonal slice is displayed and the corresponding file saved for later display in presentation programs. The file contains point coordinates in the selected plane, the velocity, and the number of rays affecting that point. The horizontal point coordinates are the distances from line A. These files can be read by Surfer and, with changes in the header lines, by other 2D contouring programs.

To examine the residuals, click **Residuals, Open residuals file**. Select the desired residuals file and click **Open**. The residuals file is shown as a line by line copy in a scrollable window. You can examine the residuals file in the scrollable window before clicking **OK**.

The residuals file contains the RMS residual for each iteration and the residual for each raypath for the last iteration. When curved-ray iterations follow straight-ray iterations, the RMS residual often increases somewhat at the first curved-ray iteration. This increase is caused by the straight-ray iterations making corrections in the wrong places, and by the ability of curved-ray iterations to find shorter-time raypaths. With additional curved-ray iterations, the process generally resumes converging to a solution (i.e., residuals tend to decrease).

To plot the residuals, click **Residuals, Plot**, and a dialog box with plotting options will appear in the upper left corner of the screen. The options for examining residuals are discussed in the chapter "Plots and Files from Residuals Files."

The 3DCylSynth2Defects data set is more suitable than the Demo1 data set for practice with the 3D cylinder display. 3DCylSynth2Defects.3dd is a synthetic data set in m and ms simulating a concrete cylinder with a low-velocity zone on the surface at X and Y = 0, Z = 4.75 to 5.25 m and another near the center at Z = 14.75 to 15.25 m. 3DCylSynth2Defects\_1Str7Bend.3dm is the calculated velocity model file using 3DCylSynthUnif3.3dm as the initial model file. The user can follow the procedure described above to run GeoTomCG and can try the various **3DCyl** display options described in the chapter "Displaying Tomograms and Saving Output Files."

## WAVEFRONT CONSTRUCTION

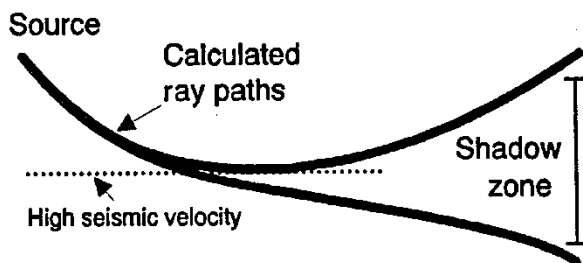
GeoTomCG provides wavefront construction as an optional alternative to ray-bending for curved-ray tracing. Much of the following discussion is adapted from the U.S. Bureau of Mines user's manual for the tomographic program MIGRATOM (Jackson and Tweeton, 1994). That program constructs wavefronts in 2D. GeoTomCG allows the wavefront construction to be performed in any combination of vertical planes. Thus, it can read and process 3D data sets, though the wavefronts are always constructed in vertical planes. The wavefronts are constructed starting at each source, so it is most efficient when there are fewer sources than receivers and the receivers are in vertical planes. Other configurations may yield much slower execution.

### Use of wavefront construction in GeoTomCG

#### Reason this option is offered

Wavefront construction is offered as an option because it will give more reliable results than ray bending where there is a region of uniform velocity overlaying a different region of a higher velocity and the straight-line raypaths from sources to receivers pass only through the upper uniform layer. This configuration applies to refraction, if a constant velocity in layers is assumed. If the source and receivers are only on the surface, then the raypaths in the uniform velocity region will not be bent by the ray-bending method, and the raypaths will not be correct. However, for most inversions, ray bending will be faster and give results that are similar to wavefront construction. Ray bending works well for crosshole inversions with an initial uniform velocity if the raypaths give good coverage over the grid. The first iteration changes the velocity distribution and creates velocity gradients that provide appropriate ray bending in the subsequent iterations.

Wavefront construction is offered rather than ray shooting and grid point-to-point ray tracing that can, in principle, handle the refraction velocity configuration. Ray shooting methods (Bregman et al., 1989) find the path to a particular receiver by tracing rays with different takeoff angles from the source, and interpolating until one passes sufficiently close to the receiver. The USBM's second-generation tomographic program, BOMCRATR, performed curved-ray calculations using the shooting method (Tweeton et al., 1994). Ray shooting methods may fail



for complicated velocity distributions, where ray trajectories can be an extremely sensitive and irregular function of takeoff angle. An example is the inability of shot rays to reach certain receivers in high-velocity areas because of refraction of the rays away from the receiver (see figure 1).

Figure 1. The "shadow-zone" problem. Rays are refracted away from high-velocity layer. Ray exit points are an extremely sensitive function of takeoff angle, causing problems for ray-shooting methods.

The grid point-to-point method (Moser, 1991), one of the ray-tracing options in the USBM program 3DTOM (Jackson and Tweeton, 1996), finds the shortest-time raypaths along grid points. These raypaths are jagged unless small voxels are used, and the traveltimes for such raypaths is usually longer than for the true shortest-time raypath.

#### How to use the wavefront construction option

GeoTomCG uses the wavefront construction option if the number of wavefront iterations is set to a number larger than 0. Wavefront parameters can be set by the user in the menu **Inversion Parameters**. The number of wavefront iterations is entered in a text box, similar to entering the number of straight-ray or ray-bending iterations. GeoTomCG will do any straight-ray iterations first, then ray-bending, then wavefront construction. To perform forward modeling with only wavefront construction, set the number of straight-ray and ray-bending iterations to 0, and set the number of wavefront iterations to 1.

The user can set the length of time steps for the wave-front iterations in the same dialog box. The Huygens algorithm is a numerical integration method applied to an initial value problem, so the accuracy depends on proper selection of the step size. The default step size is the length of time required to travel half the shortest edge of a voxel at the higher of these two velocities: (a) the highest initial point velocity, or (b) the highest straight-line source-to-receiver velocity. For sharply varying velocity models, it may be necessary to reduce the step size to obtain accurate results. The step size may be increased slightly to reduce execution time, but caution is advised: If the time steps are too large, the wavefront may fail to sample certain voxels, which will cause erroneous results.

The user should examine the plot of residuals to check for unusually large errors and should examine the display of the raypaths to check for unrealistic raypaths. If no raypath connection can be made between the source and a receiver, the calculated traveltimes and distances for that raypath is set to 0. Thus, a raypath with a calculated traveltimes of 0 indicates a problem with the wavefront construction. A shorter time step or an expanded grid volume may cure the problem.

## Method of wavefront construction in GeoTomCG

GeoTomCG, like MIGRATOM, implements wavefront construction by direct application of Huygens' Principle. Originally formulated in optics, Huygens' Principle enables modeling of the propagation of a wavefront through a heterogeneous velocity medium by treating each point on the wavefront as an instantaneous point source of wave energy. The position of the wavefront an infinitesimal time later is the locus of constructive interference of these Huygens' wavelets.

An initially infinitesimal circular wavefront at the seismic source is advanced in a series of time steps, by constructing circular wavelets about points on the wavefront, each with a radius proportional to the local velocity. The focus of constructive interference is approximated by lines tangent to adjacent Huygens wavelets. Each point on the current wavefront is advanced for the next time step to a position on its wavelet midway between tangent points (see figure 2). The trajectory of an individual tracking point on the wavefront represents a ray path. As the wavefront expands, new tracking points are added by interpolation. In the interest of computational efficiency, wavefront points are deleted when they have traveled a certain distance outside the region of interest.

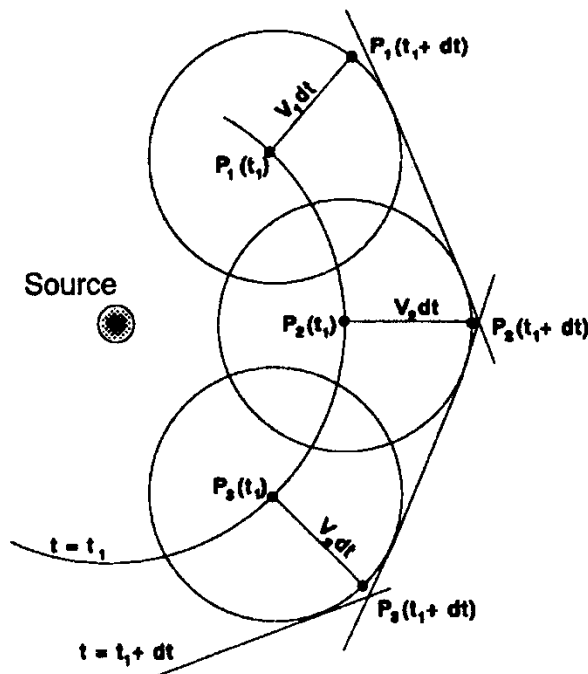


Figure 2. – Schematic illustration of Huygens' Principle.



### Initiation and propagation

The wavefront is initially defined by 360 points at  $1^\circ$  angular intervals, and at a radial distance from the source equal to  $v(S) \Delta t$ , where  $v(S)$  is the velocity at the source location and  $\Delta t$  is the length of a time step. The takeoff angle  $\phi$  is defined such that  $\phi$  equals zero for a ray traveling in the positive  $z$  direction.

At each point on the wavefront, a circular wavelet is constructed. For the  $k$ th point, the radius  $r_k$  of the wavelet is  $v(x_k, z_k) \Delta t$ , where  $v(x_k, z_k)$  is the velocity at the point. A line tangent to circles  $k$  and  $k-1$  intersects circle  $k$  at the point  $(x_k', z_k')$  (figure. 3). The coordinates of the intersection point are calculated as follows:

The distance  $d$  separating points  $k$  and  $k-1$  is equal to

$$d = \sqrt{(x_k - x_{k-1})^2 + (z_k - z_{k-1})^2} .$$

The orientation angle  $\beta$  of segment  $d$ , measured clockwise from the  $-z$  axis, is given by

$$\beta = \tan^{-1} \left[ \frac{x_k - x_{k-1}}{z_k - z_{k-1}} \right] ,$$

and the angle  $\alpha$  between  $d$  and  $r_{k-1}$  is

$$\alpha = \cos^{-1} (\Delta r / d) ,$$

where  $\Delta r = r_{k-1} - r_k$ .

Defining  $\gamma$  as  $180^\circ - (\alpha + \beta)$ , the intersection point can be calculated as

$$x_k' = x_k + r_k \sin \gamma$$

$$z_k' = z_k + r_k \cos \gamma .$$

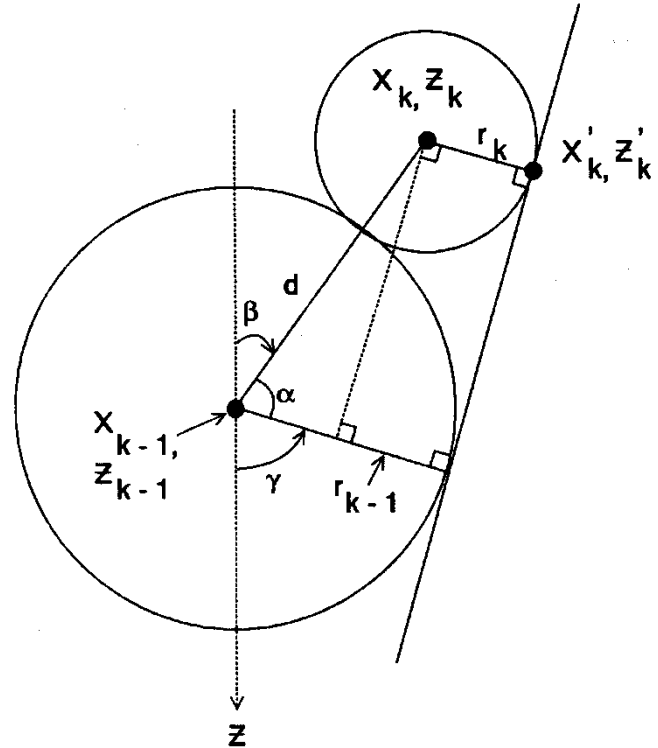


Figure 3.-Forward time step, two adjacent points. Locus of constructive interference is approximated by a line tangent to adjacent Huygens wavelets.

Repeating the process for circles  $k$  and  $k + 1$  produces another intersection point on circle  $k$ , generally different from the first intersection point. The travel angle  $\gamma'$  for the ray from point  $k$  is determined by bisection of the angle between the two intersection points (figure 4). Together with  $r_k$ ,  $\gamma'$  provides an estimate of the location of point  $k$  for the subsequent time step:

$$\hat{x}_k = x_k + r_k \sin \gamma'$$

$$\hat{z}_k = z_k + r_k \cos \gamma' .$$

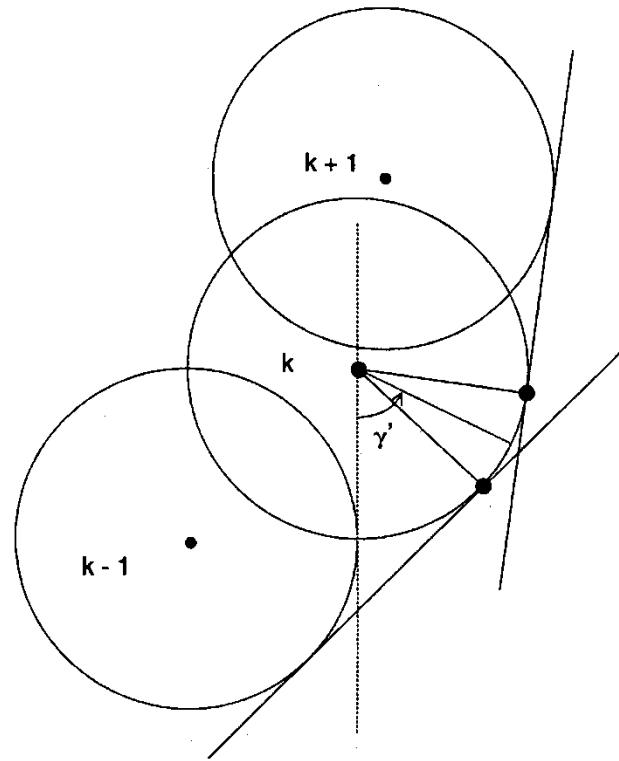


Figure 4.-Forward time step, three neighboring points. For each wavelet there are tangent points associated with two neighboring wavelets. The ray-path orientation at point  $k$  bisects these two tangent points.

### Arrivals of wavefronts at receivers

When the new location of a wavefront point has been calculated, it is necessary to determine whether the wavefront segment has crossed any receiver locations. GeoTomCG first makes a rapid test to eliminate wavefront points far from a receiver. Points that are not eliminated by the rapid test are then checked for arrivals in the following way. GeoTomCG first calculates difference vectors corresponding to the four sides of a box defined by the locations of wavefront points  $k$  and  $k - 1$  at times  $t$  (the current time step) and  $t + \Delta t$  (the succeeding time step) (see figure 5). The difference vectors  $\mathbf{A}$ ,  $\mathbf{B}$ ,  $\mathbf{C}$ , and  $\mathbf{D}$  define a counterclockwise circuit, tip-to-tail around the box. For each receiver  $R_i$ , a second set of difference vectors  $\mathbf{a}$ ,  $\mathbf{b}$ ,  $\mathbf{c}$ ,  $\mathbf{d}$  is calculated by subtracting the coordinates of the box corners from those of the receiver. Finally, the cross-products  $\mathbf{A} \times \mathbf{a}$ ,  $\mathbf{B} \times \mathbf{b}$ ,  $\mathbf{C} \times \mathbf{c}$ , and  $\mathbf{D} \times \mathbf{d}$  are calculated. If the components orthogonal to the plane of figure 5 of all four cross-products are negative, the receiver location must be within the box. Thus the wavefront segment defined by points  $k$  and  $k - 1$  sweeps across receiver  $R_i$  between time  $t$  and  $t + \Delta t$ , and an arrival is recorded.

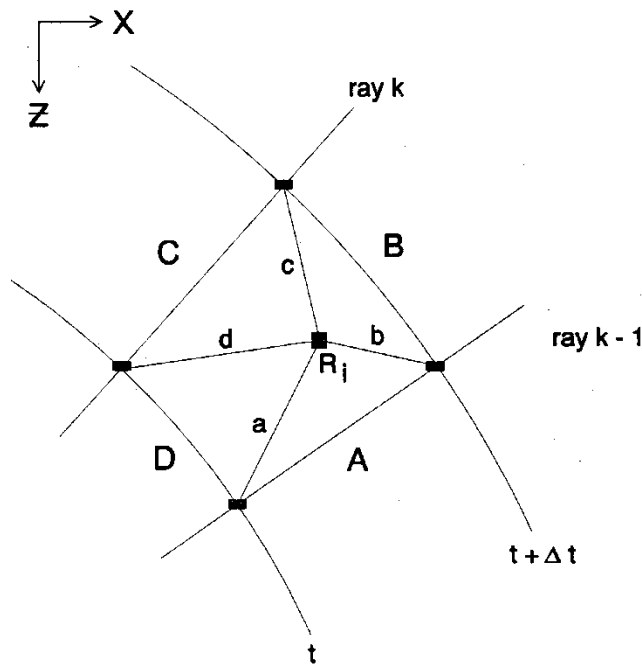


Figure 5.-Box constructed for arrival test.

Once an arrival is recognized in this way, its takeoff angle and arrival time must be calculated. GeoTomCG calculates both values by linear interpolation:

$$t_a = t + (\mathbf{a} \cdot \mathbf{A} / |\mathbf{A}|^2) \Delta t$$

$$\phi_a = \phi_{k-1} + (\mathbf{a} \cdot \mathbf{D} / |\mathbf{D}|^2) (\phi_k - \phi_{k-1}) .$$

Testing for arrivals in this manner requires much processing time. For each of the wavefront points, GeoTomCG must calculate eight difference vectors and four cross-products for each receiver. The preliminary rapid test reduces the total time required for checking for arrivals.

A possible problem that may develop in association with sharp ray curvature is a raypath leaving the grid volume before it arrives at a receiver. If a raypath leaves the grid volume, the time and path length are set to 0. This is necessary to enhance performance and to prevent memory overflows. Expanding the grid volume may solve the problem, but in some instances the wavefront method will fail anyway. However, the author has not encountered a velocity pattern where both the raybending and wavefront methods fail.

## REFERENCES

- Bregman, N. D., R. C. Bailey, and C H. Chapman, 1989, Crosshole Seismic Tomography. *Geophysics*, v. 54, pp. 200-215.
- Devaney, A. J., 1984, Geophysical Diffraction Tomography. *IEEE Trans Geosci. and Remote Sensing*, v. GE-22, 1984, pp. 3-13.
- Jackson, M. J., M. J. Friedel, D. R. Tweeton, D. F. Scott, and T. Williams, 1995, Three-dimensional Imaging of Underground Mine Structures Using Seismic Tomography. Paper in Proceedings of the Symposium on the Application of Geophysics to Engineering and Environmental Problems (SAGEEP95) (Orlando, FL, Apr. 23-26) pp. 221-230.
- Jackson, M.J., and D.R. Tweeton, 1994, MIGRATOM – Geophysical Tomography Using Wavefront Migration and Fuzzy Constraints, USBM RI 9497, 35 pp.
- Jackson, M.J., and D.R. Tweeton, 1996, 3DTOM: Three-Dimensional Geophysical Tomography, USBM RI 9617, 84 pp.
- Jessop, J. A., D. L. Boreck, M. J. Jackson, D. R. Tweeton, and M. J. Friedel, 1994, Evaluation of Stope Leaching Site Using Geotomography. Paper in In Situ Recovery of Minerals II (Santa Barbara, CA, Oct. 25-30, 1992). Engineering Foundation, (published 1994) New York, pp. 599-616.
- Lehmann, B., 2007, Seismic Traveltime Tomography for Engineering and Exploration Applications. EAGE Publications bv, DB Houton, the Netherlands.
- Lytle, R.J, K.A. Dines, E.F. Laine, and D.L. Lager., 1978, Electromagnetic Cross-Borehole Survey of a Site Proposed for an Urban Transit Station. UCRL-52484, Lawrence Livermore Laboratory, University of California, 19 pp.
- Menke, W., 1984, The Resolving Power of Cross-borehole Tomography. *Geophys. Res. Lett.*, v. 11, pp. 105-108.
- Moser, T. J., 1991, Shortest Path Calculation of Seismic Rays. *Geophysics* v. 56, pp. 59-67.
- Nabighian, M. N. (ed.) 1988, *Electromagnetic Methods in Applied Geophysics-Vol. 1, Theory*. Soc. of Exploration Geophysicists, Tulsa, OK, 513 pp.
- Peterson, J. E., B. N. P. Paulson, and T. V. McEvelly. 1985, Applications of Algebraic Reconstruction Techniques to Crosshole Seismic Data. *Geophysics*, v. 50, pp. 1566-1580.
- Santamarina, J. C, and D. Fratta. 1998, *Introduction to Discrete Signals and Inverse Problems in Civil Engineering*. ASCE Press, Reston, VA. 327 pages.
- Thomsen, L. 1986, Weak Elastic Anisotropy, *Geophysics*, v. 51, pp. 1954-1966.

Thomsen, L. 2002, Understanding Seismic Anisotropy in Exploration and Exploitation. 2002 Distinguished Instructor short Course, Distinguished Instructor Series, No. 5, Soc. of Exploration Geophysicists, Tulsa, OK, pp. 1-27 to 1-36.

Troncke, J., D.R. Tweeton, P. Dietrich, and E. Appel, 2001, Improved Crosshole Radar Tomography by Using Direct and Reflected Arrival Times. *Applied Geophysics*, v. 47, pp. 97-1005.

Tweeton, D.R., M.J. Jackson, and K.S. Roessler, 1992, BOMCRATR – A Curved Ray Tomographic Computer Program for Geophysical Applications, USBM RI 9411, 39 pp.

Um, J. and C. Thurber, 1987, A Fast Algorithm for Two-Point Seismic Ray Tracing. *Bull. Seismol. Soc. Am.*, v. 77, pp. 972-986.

## INDEX

Labels on buttons in dialog boxes and other commands are listed in this font.  
Other terms are listed in this font.

### 2

2D slice plane, 42

### 3

3D Cyl, 45

3D Rect, 43

### A

**Adjust display colors**, 43

amplitude processing, examples, 70

anisotropy, 25

**Anisotropy for points**, 48

anisotropy, checking for, 54

Attenuation tomography, 62

### C

color spectrum, adjust, 47

**common-receiver gathers**, 20

**common-source gathers**, 20

constraints, 34

**Constraints for points**, 48

curved-ray iterations, 37

### D

**Damping**, 39

**Data, Plot**, 19

**Data, Remove outliers**, 23

**Difference of Model1 and Model2**, 61

### E

**Edit, Copy**, 48

**Export 3D SlicerDicer File**, 50

**Export 3D Tecplot File**, 50

### G

**Generate calculated data file**, 53

**Green Red spectrum**, 43

grid constraints, 35

### M

**Model, Compare**, 61

**Model, Display model**, 26

**Model, Edit model**, 27

**Model, Export 3D SlicerDicer File**, 50

**Model, Export 3D Tecplot File**, 50

**Model, Extract slice, hor X axis**, 49

**Model, Extract slice, hor Y axis**, 49

**Model, Extract vertical slice**, 49

**Model, Generate model**, 24

### O

**Options, Inversion parameters**, 37

**Overlay Surfer BLN file?**, 47

### P

Point constraints, 34

### R

**Ratio of Model1/Model2**, 61

ray bending, 12

**Raypaths, Export 2-D ray file**, 51

**Raypaths, Save Raypaths**, 40

**Rays per grid point**, 48

**Rays per point**, 46

**Red Blue spectrum**, 43

reflected rays, 36

residuals file, 41

**Residuals, Open residuals file**, 52

**Residuals, Plot**, 52

**Residuals, Remove outliers in measured data**, 53

**Rotate Coordinates**, 23

rotation, 44

**Run curved-ray attenuation**, 40

**Run standard inversion**, 40

**S**

**Show sources and receivers**, 46

SIRT calculations, 12

**Smoothing**, 38

straight-ray iterations, 37

**T**

Thomsen anisotropy parameters, 39

tilt, 44

tutorial with demo data, 74

**V**

**Velocity at grid points**, 48

velocity models, differences or ratios, 61