UNITED STATES DEPARTMENT OF THE INTERIOR
GEOLOGICAL SURVEY
345 Middlefield Road / MS977
Menlo Park, CA 94025
(415) 329-4763
FTS 459-4763

User's Manual for

RAY84 / R83PLT

Interactive Two-Dimensional Raytracing/Synthetic Seismogram Package

By

James H. Luetgert

15 March 1988

OPEN-FILE REPORT 88-238

This report is preliminary and has not been reviewed for conformity with U.S. Geological Survey editorial standards.
Any use of trade names is for descriptive purposes only and does not imply endorsement by the U.S.G.S.

Menlo Park, California
1988

Table of Contents

Introduction 1

RAY84 Input 3

- Run Flags and Plotting Specifications 3
- Specification of the Model 9
  - Interfaces 10
  - Velocities 11
- Ray Sets 16

Interactive Use of RAY84 24

- Variable Changing Commands 24
- General Commands 30
- Summary of Interactive Commands 36

Output from RAY84 38

Plotting Observed Travel Time Data 39

Modeling Diffractions with RAY84 42

Running RAY84 in Batch Mode 44

R83PLT 46

References 52

Interactive Two-Dimensional Raytracing/Synthetic Seismogram Package (VAX/VMS version)

Introduction

The package consists of two programs: RAY84 and R83PLT. RAY84 shoots rays through the user's velocity model. A simple parameter change then causes the program to calculate the complex amplitude and geometric spreading of these rays. R83PLT then operates on an output file from RAY84 to produce synthetic seismograms for the modeled structure.

RAY84 may be run interactively from a VT100 compatible graphics terminal or exclusively from a control file. Due to the interactive features of RAY84, it can only by used on VT100 compatible terminals with graphics capabilities when in interactive mode.

Models are defined by two or more interfaces extending across the model from left to right. Interfaces may "pinch out" but may not cross. Any pair of successive interfaces describe a layer, within which the velocity may be defined in terms of the velocity at the top and bottom of the layer. Within any layer the velocity may be inhomogeneous, but continuous. First or second order discontinuities in velocity occur at interfaces. The ray tracing algorithm used calculates the propagation of rays within a layer by the stepwise integration of the system of first order differential equations,

$$\frac{dX(t)}{dt} = V(X,Z)\,\mathrm{Sin}\,(\theta)$$

$$\frac{dZ(t)}{dt} = V(X,Z)\,\mathrm{Cos}\,(\theta)$$

$$\frac{d\theta(t)}{dt} = \frac{dV}{dX}\,\mathrm{Cos}\,(\theta) - \frac{dV}{dZ}\,\mathrm{Sin}\,(\theta)$$

where θ is the ray's angle from the vertical.

By supplying a definition of V(X,Z) and initial values for X, Z, t, and θ, subsequent values of X, Z, t, and θ may be calculated by simultaneously integrating the above three equations over small steps in time. For derivation and details see Cerveny *et al.*, (1977).

Lithologic interfaces are represented in the model as first or second order velocity discontinuities. When an interface is encountered in the calculation of a ray, Sneil's law is applied and the calculation is continued.

RAY84 may be run as a batch process, as a file-controlled process from a VT100 compatible terminal, or as an interactive process from a VT100 compatible terminal. Model development is normally done as a combination of interactive ray tracing and use of the automatic branch finding features.

The graphical output of RAY84 consists of a plot of the model with rays and a travel time plot with optional observed and calculated arrival times. You may choose either, both, or neither of these plots via a control flag (JFLG(3)).

A typical session of working with the package might run as follows. First, make changes to the structure as determined from the last session. Next run RAY84 interactively until the desired sets of rays are obtained. You may keep track of ray angles, etc. on a pad of paper or use the interactive SAVE function to save them in a file. Exit the program, edit the model and trace again until you are satisfied with the arrival times. Exit the program a final time and edit your chosen ray sets into the control file. Run the program non-interactively one final time to make sure everything is correct. When satisfied with this step, change a flag in the input file to RAY84 and run the program again. This time the amplitudes are calculated and written to a disc file. To calculate the synthetic seismograms, you must edit some parameters which govern the plotting which have been added to the file containing the amplitudes by RAY84. R83PLT is then run on this file. The synthetics are then compared to the real data and the process begins again.

This manual begins with a discussion of the input control file for RAY84, followed by a description of the interactive command set. Finally, the input control file for R83PLT is described.

RAY84 Input

Input to RAY84 consists of a formatted control file containing three sections of information plus an optional auxiliary file containing the observed travel times and ranges to be plotted with the calculated travel times.

Section I - Run Flags and Plotting Specifications

This section of the file may be preceded by any number of comment lines containing an asterisk (*) in the first column. The information in this section controls the program's output to files and the appearance of the plotted output.

Line 1 - Title (80A1).
The title is printed in the run header of the output list and appears on the plot.

Line 2 - JFLG (10I1) Flags to be in effect for the entire run.

JFLG(1) - Print flag for input data.
- =0 Print the header and input file name only.
- =1 Add the graphics specifications.
- =2 Add the model input.
- =3 Add the model analysis.

JFLG(2) =0 Not currently in use.

JFLG(3) - Plot flag.
- =0 Plot travel time axis and model.
- =1 Plot model only.
- =2 Plot travel time axis only.
- =3 No plot.

JFLG(4) - The summary file provides a tabulation of ranges and times for all rays in a format compatible with that of observed travel time data files, The summary file is used by other programs for plotting calculated arrival times.
- =0 Do not dump ray summary to a summary file.
- =1 Dump ray summary to a summary file.
- =2 Dump ray summary to RAY.SUM.

JFLG(5) - Measurement units.
- =0 Scaling factors defined in terms of inches.
- =1 Scaling factors defined in terms of cm.

JFLG(6) - Observed data.
- =0 No plot of observed travel time data.
- =1 Program will request the name of a file containing travel time data and plot it.

JFLG(7) - Amplitude calculation for R83PLT.
- =0 No computation of amplitudes.
- =1 Compute amplitudes and write to an amplitude file.
- =2 Compute amplitudes and write to RAY.AMP.

JFLG(8) - Not currently in use.

JFLG(9) - Not currently in use.

JFLG(10)- Display and plot output.
- =0 Assume that plotted output will be to the VT100 only and skip the initial plot package prompts. The scaling factors for the axes specified on lines 3-5 will be ignored and the lengths of the axes will be independently re-computed to fill the screen.
- =1 Assume that plotted output will be to the VT100 only and skip the initial plot package prompts. The plot will be increased or decreased in overall size to fit the screen.
- =2 Use normal plotting prompts. The plot will be increased or decreased in overall size to fit the VT100 screen. If a BATCH.PLT vector file is created, it will have the size specified by lines 3-5 below. See the list output file RAY.OUT for the size specifications required to run VIEWR.
- =3 Use normal plotting prompts. The scaling factors for the axes specified on lines 3-5 will be ignored and the lengths of the axes will be independently re-computed to fill the screen. If a BATCH.PLT vector file is created, it will be small enough to fit on a VERSATEC sheet without scaling.

When you are first starting to model a structure you will probably want to set the JFLGs to 3000010000 or 3000110000. Once you are satisfied that the model is behaving properly, you may want to reduce the size of JFLG(1) to make the list output file smaller.

Lines 3 and 4 define the distance and depth axes for the model plot. The values given for RMIN, RMAX, ZMIN, and ZMAX need not correspond to the bounds of the defined velocity model. Thus, you may work on small sections of a velocity model at a time. This is particularly useful for working on the near-surface portions of models near shot points.

Line 3 - RMIN, RMAX, RSF, DR, NSR, RBIAS (4F10.0, I10, F10.0)
Range axis plot specifications.

RMIN = Minimum range (kilometers).

RMAX = Maximum range (kilometers).

RSF = Range scale factor (in/km or cm/km).

DR = Distance between labeled tics (kilometers).

NSR = Number of small divisions between labeled tics.

RBIAS = Bias to be applied to the range axis labeling. This may be used to make the range axis reflect distance from a shot point. This variable only affects the axis labeling. Subsequent references to locations within the model, such as position of the source, are given in absolute model coordinates.

Line 4 - ZMIN, ZMAX, ZSF, DZ, NSZ, ZBIAS (4F10.0, I10, F10.0)
Depth axis plot specifications.

ZMIN = Minimum depth (kilometers).

ZMAX = Maximum depth (kilometers).

ZSF = Depth scale factor (in/km or cm/km).

DZ = Distance between labeled tics (kilometers).

NSZ = Number of small divisions between labeled tics.

ZBIAS = Bias to be applied to the depth axis labeling. This may be used to shift the zero point on the plot to sea level for publication. This variable only affects the axis labeling. Subsequent references to locations within the model, such as position of the source, are given in absolute model coordinates.

Line 5 - TMIN, TMAX, TSF, DT, NST, (4F10.0, I10)
Time axis plot specifications.

TMIN = Minimum time (seconds).

TMAX = Maximum time (seconds).

TSF = Time scale factor (in/sec or cm/sec).

DT = Distance between labeled tics (seconds).

NST = Number of small divisions between labeled tics.

Line 6 - RV, HTC, HID, HSYM, HOBS (5F10.0)
Plot specifications.

RV = Reducing velocity in km/sec (use 0.0 or 10000.0 for an unreduced plot).

HTC = Size of tics and axis labels in inches or cm. An entry of 0.0 sets HTC to a default size of 0.3. A negative entry sets HTC to zero and eliminates tics and axis labels.

HID = Size of the plot ID in inches or cm. An entry of 0.0 sets HID to a default size of 0.3. A negative entry sets HID to zero and eliminates the plot ID.

HSYM = Size of the calculated arrivals plotted on the travel time plot in inches or cm. An entry of 0.0 sets HSYM to a default size of 0.2. A negative entry sets HSYM to zero and eliminates the symbols from the travel time plot.

HOBS = Size of the observed arrivals plotted on the travel time plot in inches or cm. An entry of 0.0 sets HOBS to a default size of 0.2. A negative entry sets HOBS to zero and eliminates the symbols from the travel time plot.

Line 7 - ROSER, ROSET, ROSES, ROSIGN (4F10.0)
Velocity rosette specifications.

ROSER = Range of center of velocity rosette (kilometers).

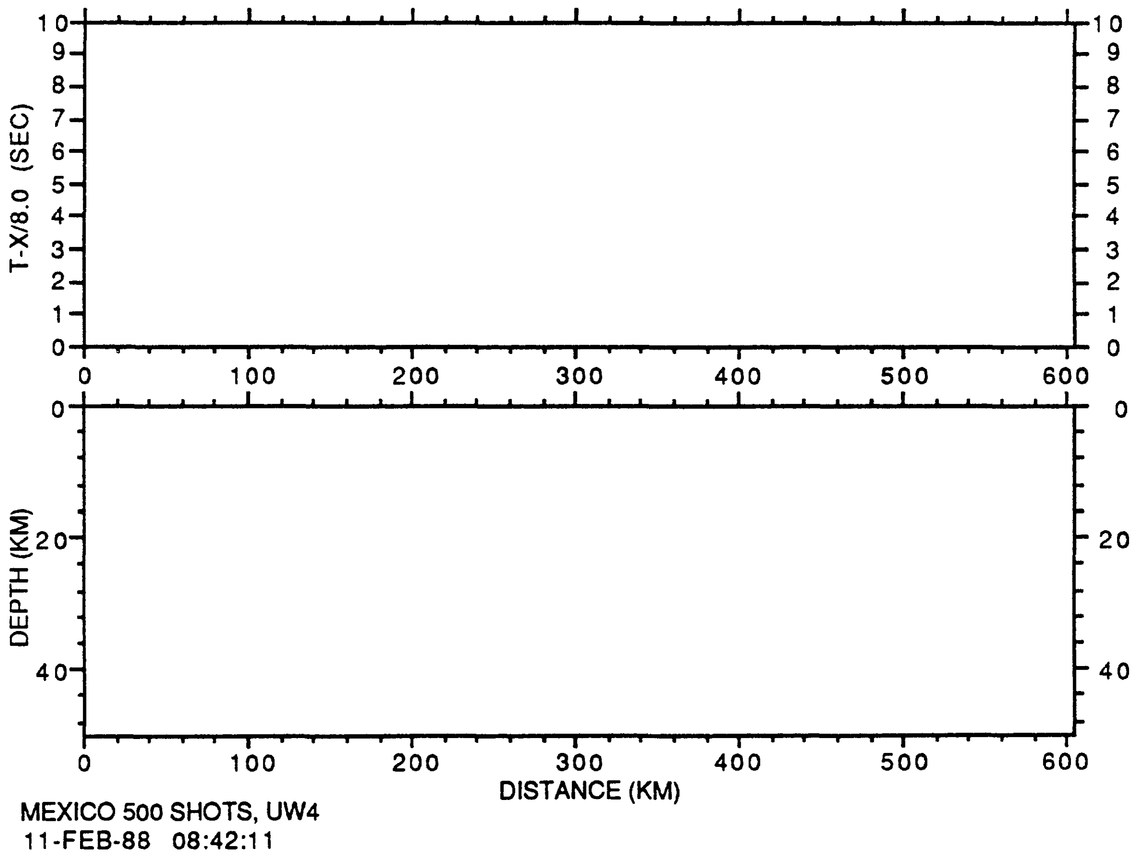
ROSET = Time for center of velocity rosette (seconds).

ROSES = Length of arms of velocity rosette in inches or cm. Set equal to zero for no rosette.

ROSIGN = Direction of the velocity rosette. +1.0=right; -1.0=left; 0.0=both.

The following is an example of a control file and the axes generated by it to show how the first section of information should look.

```
*
*-- RUN FLAGS AND PLOTTING SPECS.
*
MEXICO 500 SHOTS; UW4
3000100000
     0.000    605.000     0.040   100.000         5      0.000
     0.000     50.000     0.100    20.000         5      0.000
     0.000     10.000     0.500     1.000         1
      8.00       0.30      0.30      0.20      0.20
     70.00      5.000     0.000     0.000
*
*-- VELOCITY MODEL.
*
$OF.M84
*
*-- RAY SETS.
*
****
```



Section II - Specification of the Model.

The definition of the model may reside in a separate file. If this is the case, the first model line in the control file must have a $ in the first column, followed by the name of the model file. Placing the model definition in a separate file allows you to have a single model for a profile which is referred to by control files for each of several shot points. An additional advantage of having the model in a separate file is that it enables you to modify the model during the interactive ray trace process using standard VMS editors. To do this, first enter CNTRL/C to suspend RAY84. Invoke the editor via spawning;

```
$SPAWN/NOLOG EDT MODEL.M84
```

and edit your model file (here called MODEL.M84). After exiting the editor, type CONTINUE to resume operation of RAY84. To make RAY84 read your revised model, type MODEL.

The model is specified in two steps. First, interfaces are defined. Then velocities at the interfaces are defined.

II-A - Interfaces

The model is defined in terms of NBND interfaces each of which must extend from the left edge of the model to the right edge. The interfaces define NLAY = NBND-1 layers whose velocities are defined in the second part of the model section. The first interface is commonly used to represent surface topography. Interfaces may merge but not cross. That is, a layer, defined by two adjacent interfaces, may be made to "pinch out". Care must be used in the choice of velocities, however, when pinching out layers. It is best to only pinch out layers in regions of constant velocity.

Line 1 - NBND, (NPNT(I), I=1,NBND) (14I5)

NBND = Number of interfaces (25 or fewer).

NPNT(I) = Number of (X,Z) points used to define the I^{th} interface (85 or fewer).

Line 2-N - Interface lines; each interface reads as an ascending sequence of X values, possibly requiring multiple lines, followed on a new line by a sequence of corresponding Z values under format (10F7.0). Z is defined as positive downward. Although the program will work with sea level defined as Z=0 and elevations above sea level defined as negative numbers, it is ultimately less confusing to define an artificially elevated datum such that all interfaces are defined with positive values of Z. For final plots of the model, the Z-axis may be properly labeled with respect to sea level by using the variable ZBIAS. Interfaces must extend across the entire model. All interfaces should have the same X values at the beginning and end of the model.

Groups of interface lines may be separated by comment lines containing an asterisk (*) in the first column to make the input file easier to read.

If you wish to preempt the plotting of a given interface in your model, precede that group of interface lines with a comment line reading *1 . This feature may be used to delete all interfaces except the first for plots of topography to accompany record sections. Normally, if a section of an interface represents a second order velocity discontinuity (*i.e.* the velocity is the same above and below the interface), that part of the interface is plotted as a dashed line. To plot second order discontinuities as solid lines, precede with *2 . To preempt the plotting of dashed segments of an interface, precede that group of interface lines with a comment line reading *3 . To plot the entire interface as a dashed line, precede with *4 .

Note: Concerning the interface lines, if Z_i is entered as 999.0, Z_i will be linearly interpolated between the nearest defined points.

II-B - Specification of the Velocities

The velocities are specified along vertical grid lines in the model. Where these lines intersect the layers, the velocity is specified at the top and bottom interfaces.

This subdivides the model space into a number of polygonal areas each of which has a velocity defined at four points. For each of these areas a polynomial "velocity surface" is calculated which passes through the defined points. This is a numerically neat way of dealing with specifying velocities at any point in the model, but it is the source of many problems and frustrations for the user if not properly used.

Remember that the velocity at an interface is explicitly defined only at the intersection point with a velocity net line. At all other points along the interface the velocity is the intersection of the interface with the velocity surface. In regions of the model in which there are large structural or velocity variations, the resultant surface may have substantial curvature which leads to unexpected velocities.

The solution to the large curvature problem is to break up the surface into smaller surfaces each having smaller curvature through the use of more velocity net lines. There is, however, a trade-off here in that the model becomes cumbersome to edit if you use more grid lines than you need.

Start off with grid lines at the left and right sides of the model and at those ranges where they are needed to define lateral velocity changes. Put additional grid lines in any large gaps. Examine the RAY.OUT file to see if there are any obvious velocity problems. If you see rays which do not behave as expected, consider inserting additional grid lines.

Line 1 - NXSPD, NZSPD, VFACTR, GMAX (2I5, 2F10.0)

NXSPD = Number of vertical grid lines (85 or fewer).

NZSPD = Number of horizontal grid lines (Always use NZSPD=2).

VFACTR = A multiplicative factor used for determining S-velocities in the model. This is used internally to calculate S-velocities from the P-velocity model. If VFACTR = 0.0, it is set to 1.732 . Ray paths with mixed S and P paths must be generated with the ray code (see page 15). If the first entry in the current ray code is negative indicating S-wave propagation, the S-velocity model will be used whether or not the ray code is enabled.

GMAX = Maximum vertical velocity gradient in km/sec/km. If your model specifies velocities having gradients greater than GMAX, a warning message is printed and the velocity gradient is set to GMAX. If GMAX = 0.0, it is set to 2.0 by the program.

Line 2 - SPDNET(1,1), . . ., SPDNET(1,NXSPD) (10F7.0)

SPDNET(1,I) = X-coordinate for Ith vertical grid line. These need not be in ascending order, but the following velocity values should follow the same ordering scheme. The minimum and maximum X-values should correspond to the minimum and maximum X-values of the interfaces.

Line 3 - SPDNET(2,1), . . ., SPDNET(2,2) (2F7.0)

SPDNET(2,1) = Z-coordinate of the top of the model (use SPDNET(2,1) = 0.0).

SPDNET(2,2) = Z-coordinate of the bottom of the model.

Line 4-N - Velocities (10F7.0)

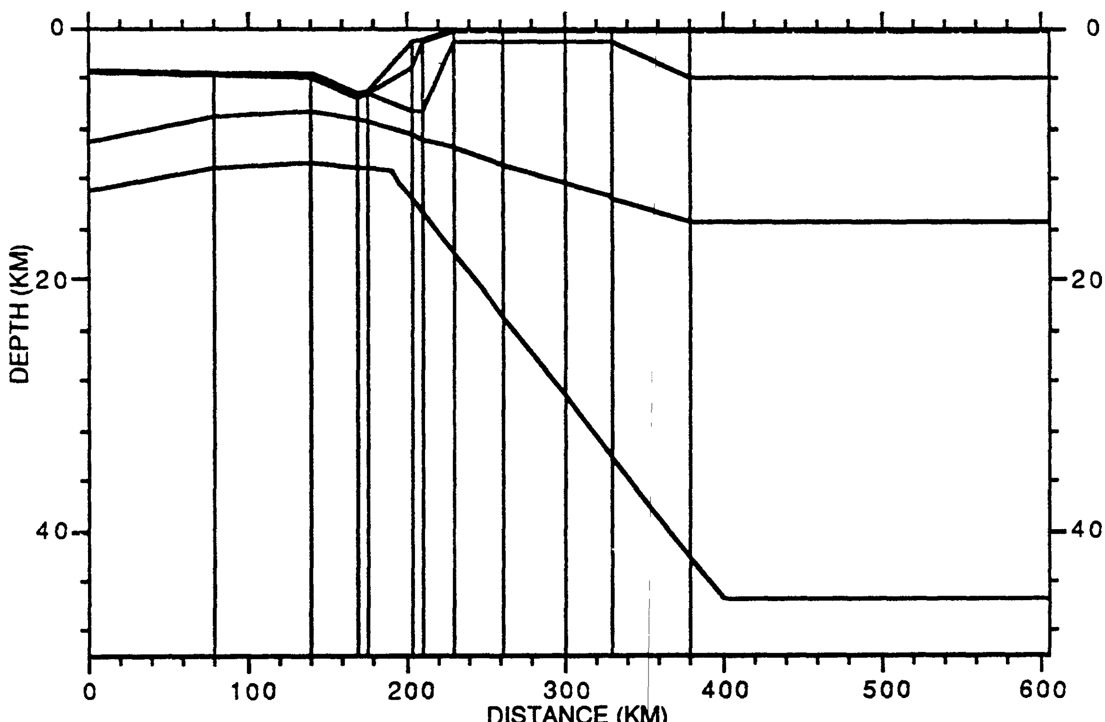
A block of lines containing velocity data for intersections of the vertical grid lines with the interfaces. Velocities at the top and bottom of each layer are read in along interfaces from left to right (or in the order specified in Line 2 above). First, NXSPD velocities at the top of the layer are read. Then, starting on a new line, NXSPD velocities at the bottom of the layer are read. For any layer there are 2*NXSPD values. Layers are taken from top to bottom.

Note: If a 0.0 is entered for VEL(I), VEL(I) will be set to VEL(I-1), the previous velocity. If a negative number is entered, VEL(I) will be linearly interpolated between the nearest defined velocities.

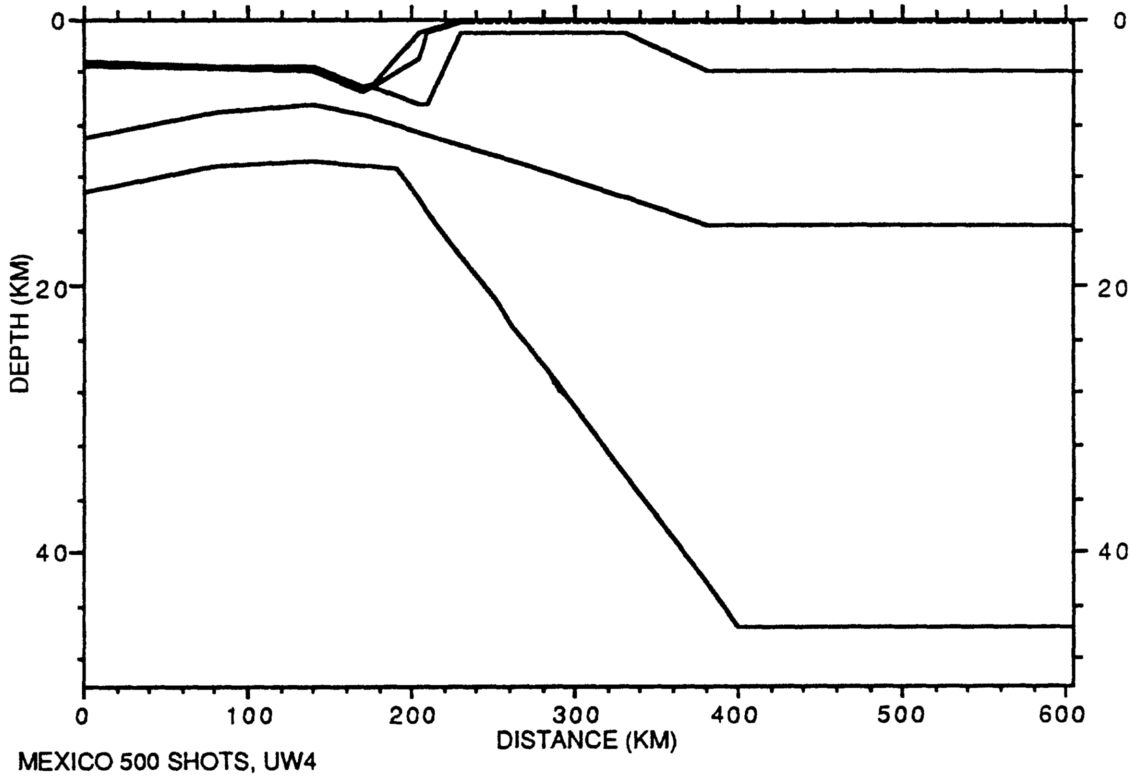
The following is an example of a model specification followed by a figure showing the model with velocity grid lines drawn and without grid lines as the model will appear in RAY84.

```
* MEXICO SUBDUCTION MODEL
* CONTAINS A HIGH VELOCITY AREA BEHIND THE SIERRA
* MADRE DEL SUR
    6   14   14   14   14   13    2
*       INTERFACE     1
     0.     80.    110.    140.    170.    176.    204.    210.    230.    262.
   330.    380.    500.    605.
    3.2     3.5     3.5     3.6     5.2     5.0     1.0     0.9     0.1     0.1
    0.1     0.1     0.1     0.1
*       INTERFACE     2
     0.     80.    110.    140.    170.    176.    204.    210.    230.    262.
   330.    380.    500.    605.
    3.4     3.7     3.8     3.8     5.5     5.1     3.0     1.0     0.2     0.2
    0.2     0.2     0.2     0.2
*       INTERFACE     3
     0.     80.    140.    170.    176.    204.    210.    230.    262.    300.
   330.    380.    500.    605.
    3.5     3.8     3.9     5.6     5.2     6.5     6.5     1.0     1.0     1.0
    1.0     4.0     4.0     4.0
*       INTERFACE     4
     0.     80.   ·140.    170.    176.    204.    210.    230.    262.    300.
   328.    380.    500.    605.
    9.0     7.0     6.5     7.2    7.44    8.54    8.78    9.57   10.84   12.34
  13.44    15.5    15.5    15.5
*       INTERFACE     5
* DOWNGOING SLAB MOVED 20 KM TO THE RIGHT.
     0.     80.    140.    190.    196.    224.    230.    250.    282.    350.
   400.    500.    605.
*      0.     80.    140.    170.    176.    204.    210.    230.    262.    330.
*    380.    500.    605.
   13.0    11.1    10.7    11.3   12.28   16.84   17.81   21.07   26.28   37.36
   45.5    45.5    45.5
*       INTERFACE     6
     0.    605.
    50.     50.
   13     2    1.7320       2.0
     0.     80.    140.    170.    176.    204.    210.    230.    262.    300.
   330.    380.    605.
     0.     51.
*     LAYER      1A
*  1.500   1.500   1.500   1.500   1.500   1.500   1.500   4.100   4.100   4.100
*  4.100   4.100   4.100
*  1.500   1.500   1.500   1.500   1.500   1.500   1.500   4.200   4.200   4.200
*  4.200   4.200   4.200
*     LAYER      1
  2.000   2.000   2.000   2.000   2.000   2.000   3.500   4.300   4.300   4.300
  4.300   4.300   4.300
```

```
   2.300   2.300   2.300   2.300   2.100   2.600   3.600   4.300   4.300   4.300
   4.300   4.300   4.300
*     LAYER     2
   2.300   2.300   2.300   2.300   2.200   4.300   4.300   4.300   4.300   4.300
   4.300   4.300   4.300
   2.300   2.300   2.300   2.300   2.200   4.600   4.600   4.600   4.600   4.600
   4.600   4.600   4.600
*     LAYER     3
   5.000   5.000   5.000   5.000   5.000   5.000   5.000   6.100   6.100   6.100
   6.100   5.000   5.000
   5.700   5.700   5.700   5.700   5.700   5.700   5.700   6.200   6.200   6.200
   6.200   5.700   5.700
*     LAYER     4
   6.850   6.850   6.850   6.850   6.850   6.850   6.850   6.850   6.850   6.850
   6.850   6.850   6.850
   7.000   7.000   7.000   7.000   7.000   7.000   7.000   7.000   7.000   7.000
   7.000   7.000   7.000
*     LAYER     5
   8.200   8.200   8.200   8.200   8.200   8.200   8.200   8.200   8.200   8.200
   8.200   8.200   8.200
   8.400   8.400   8.400   8.400   8.400   8.400   8.400   8.400   8.400   8.400
   8.400   8.400   8.400
****
****
```



MEXICO 500 SHOTS, UW4
11-FEB-88 08:42:11



MEXICO 500 SHOTS, UW4
11-FEB-88 08:42:11

Section III - Ray Sets

Each group of rays to be shot is called a ray set and is defined by four lines. There may be any number of ray sets. Ray sets may be separated by comment lines containing an asterisk (*) in the first column.

Line 1 - (IFLG(I), I=1,10), TEXT (10I1, 40A1)
Flags to be in effect for the ray set.

IFLG(1) - Print flag for rayset input.
- =0 No print of rayset input.
- =1 Print starting values only.
- =2 Print all rayset parameters.

IFLG(2) - Plot flag for rays.
- =0 Plot all the rays of the set.
- =1 Plot only successful rays.
- =2 No plot of rays.

IFLG(3) - Plot flag for travel times.
- =0 Plot travel times of successful rays, connecting points which are on the same branch.
- =1 Plot travel times of rays without connecting points which are on the same branch.
- =2 Do not plot travel times.

IFLG(4) - Ray selection.
- =0 Accept all refracted and super critical rays.
- =1 Use the ray code (line 4 of ray set) to define rays.
- =2 Accept only refracted rays.

IFLG(5) - Print flag for ray information after calculation.
- =0 No print of ray information after calculation.
- =1 Print ray input only.
- =2 Print ray input and travel times.
- =3 Print ray input, travel times, and information concerning whether the ray bottomed with reflection or refraction.
- =4 Full ray dump - location and ray parameter at each time step.

IFLG(6) - Calculation flag.
- =0 Use this ray set.
- =1 Skip this ray set.

IFLG(7) - Ray termination flag. Used when the ray code is <u>not</u> in use (IFLG(4) ≠ 1).
- =0 Successful ray defined as one which returns to the surface.
- =1 Successful ray defined as one which terminates at an intermediate interface (defined on Line 3).

IFLG(8) - Automatic calculation flag. Used only when RAY84 is running as a batch process. When this flag is set, the program will attempt to compute NRAY rays for each refracted and super-critical reflection branch.
- =0 No auto calculation.
- =1 Auto calculation.

IFLG(9) - Diffraction flag.
- =0 No diffraction.
- =1 Source for the rayset will be located at a diffraction point. Reduced time will be calculated from XPOSDF on Line 3.
- =2 Source for the rayset will be located at a diffraction point. Reduced time will be calculated from the diffraction point.

IFLG(10) - Interactive flag.
- =0 Ray set is calculated normally; program proceeds to next rayset.
- =1 Following the calculation of this rayset, the program will go into interactive mode.

TEXT - Forty characters describing the ray set (optional).

Line 2 - T0, TLIM, TSTEP, TPREC, LINK, IFZ, ITYPE, TRANCT
(4F10.0, 4F5.0)

T0 = Initial time of rays in seconds. This parameter is usually 0.0 .

TLIM = Time limit on calculation. That is, if the calculated travel time of a ray is greater than TLIM, the ray is terminated.

TSTEP = Time step of integration (seconds). The position and ray parameter are calculated every TSTEP seconds. Typically this ranges between 0.5 and 0.02 . To speed up your initial modeling, start with a large value for TSTEP.

TPREC = Precision of calculation. For crustal refraction work, an appropriate value is 0.0001 .

The next four parameters only need to be entered if amplitudes are to be calculated.

LINK = An integer (the floating point number in the input file becomes truncated on input to the program) which relates amplitudes in R83PLT. In practice, each distinct travel time branch has its own value of LINK. However, sometimes it is useful to give two travel time branches the same value of LINK, such as where the user wants a wide angle reflection to blend smoothly with the corresponding refracted arrival. LINK also identifies the multiplicity:

1	< LINK < 99	for P
100	< LINK < 199	for PP
200	< LINK < 299	for PPP

This is necessary to identify the phase changes implicit in multiple reflections. The user should experiment with LINK to become familiar with its peculiarities.

IFZ
=0 The vertical component of the seismogram is calculated.
=1 The radial component of the seismogram is calculated.

ITYPE - A flag identifying the ray type for phase changes.
=0 Pure reflection.
=1 Pure refraction.
=2 Two bounces; a refraction, then a reflection.
=3 Two bounces; a reflection, then a refraction.

TRANCT - At any ray step the transmission coefficient is set to 1.0 if the velocity contrast is less than TRANCT. As a ray refracts through a gradient it is actually transmitting through numerous thin layers. The calculation of all these transmission coefficients is time consuming and for a gradient the value is always near 1.0 . These unnecessary calculations can be avoided by a proper choice of TRANCT. A value of 0.1 is a good trial value.

Line 3 - XPOS, ZPOS, ANGLE, ENDRAY, RAYN, STFLG, XPOSDF
(4F10.0, 2F5.0, F10.0) Ray set definition.

XPOS = Range of source relative to the model in kilometers.

ZPOS = Depth of source relative to the model in kilometers. If ZPOS is above the first interface, ZPOS will be placed just beneath the first interface.

ANGLE = Initial angle of the first ray in degrees from the downward vertical.
0 to 180 are starting in the +X direction (*i.e.* propagating to the right).
0 to -180 are starting in the -X direction (*i.e.* propagating to the left).

ENDRAY = Final angle of ray set. RAYN rays between ANGLE and ENDRAY are shot.

RAYN = Number of rays to be shot. (≤ 100)

STFLG = Interface stop flag - only used if IFLG(7) = 1 and IFLG(4) ≠ 1; in which case the ray terminates at the interface specified by STFLG and the travel time is computed. If STFLG > 0, the ray will stop upon encountering the interface from below; if STFLG < 0, the ray will stop upon encountering the boundary from above. Note that STFLG is disabled if a ray code is being used.

XPOSDF = Actual X-location of source in a diffraction calculation. Ignored if IFLG(9) = 0 .

Line 4 - KREF, (NREF(I), I=1, KREF) (14I5)
Ray code. The ray code is used to specify specific acceptable paths for the rays. The ray code is only used if IFLG(4) = 1 .

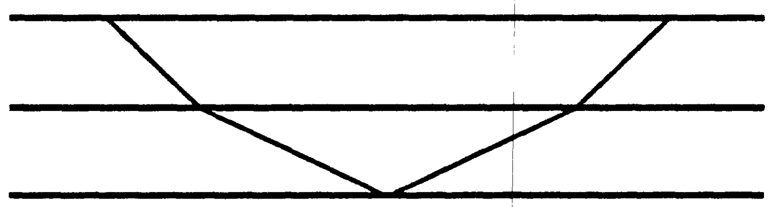
KREF = Number of entries in the ray code (≤ 100).

NREF(I) = Number of the Ith layer through which the ray travels. A negative entry indicates S-wave propagation through that layer. If the first entry of the ray code is negative, S-velocities will be used whether or not the ray code is enabled.

Examples of the ray code:

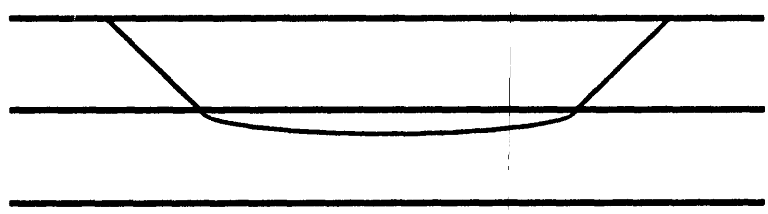
4 1 2 2 1

Reflection off the third interface.



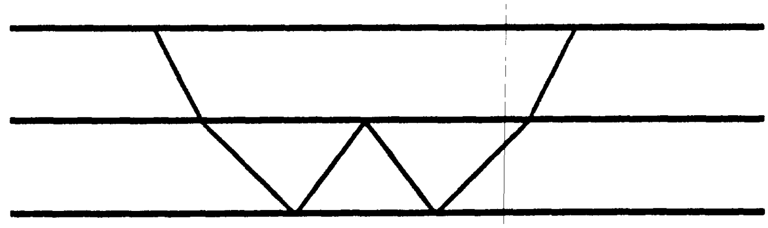
3 1 2 1

Refraction within layer 2.



6 1 2 2 2 2 1

Multiple reflection off the third interface.



If you are shooting multiple raysets, you will find that much of the information described above is redundant and makes the editing of raysets confusing. As a convenience, there are a series of abbreviated commands which may be given to update a subset of the rayset information between raysets. The abbreviated commands have an @ in the first column and are followed by a line containing the revised information. These entries are read in free format, so separate them by commas and make sure there is an entry for each variable to be read.

@ R - The following line gives ANGLE, ENDRAY, and NRAY.
@ T - The following line gives T0, TLIM, TSTEP, and TPREC.
@ C - The following line(s) give NCODE, (NREF(I),I=1,NCODE).
@ S - The following line gives XPOS, ZPOS, STFLG, XPOSDF.
@ F - The following line gives (IFLG(I),I=1,10).
@ P - Initiate calculation of a revised rayset.

Thus, a series of raysets for different starting angles might be specified,

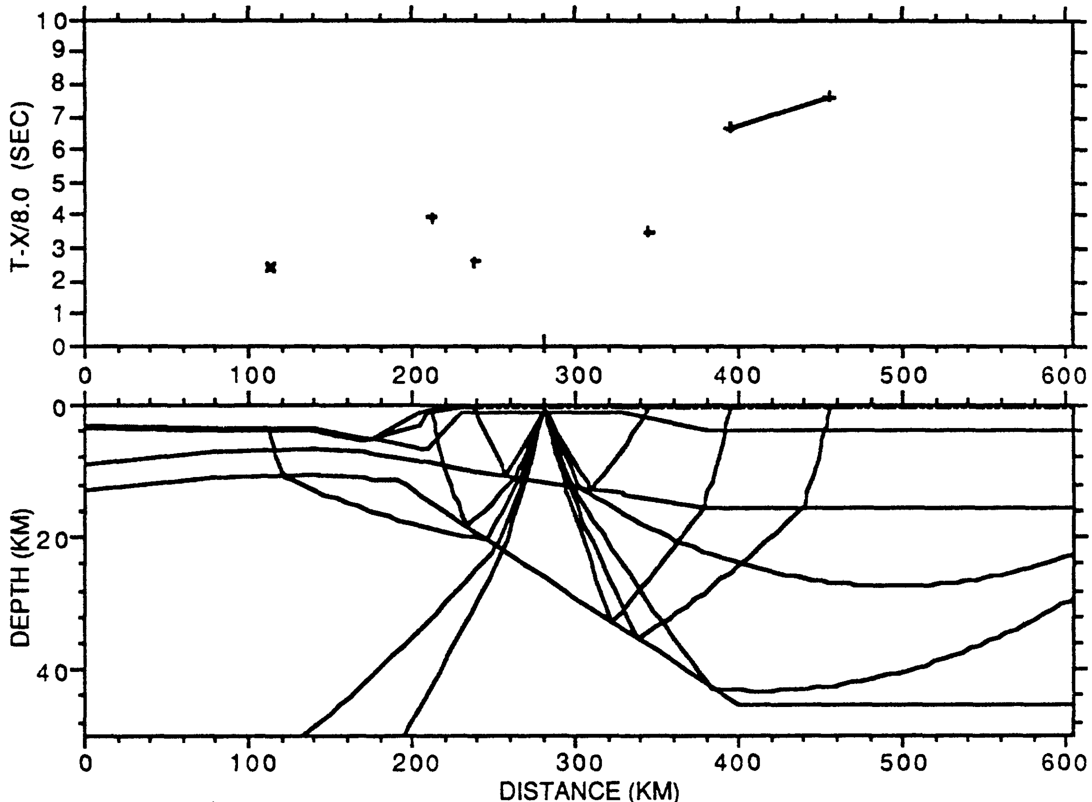
```
@R
23.1234, 32.345, 10
@P
@R
35.23, 41.23, 10
@P
```

A series of raysets using the ray code might be specified,

```
@F
1,0,0,1,2,0,0,0,0,0,
@C
6,1,2,3,3,2,1,
@R
23.1234, 32.345, 10
@P
@R
35.23, 41.23, 10
@P
```

The following example shows a complete control file and the resultant plot. The model file is the same as that used in the previous section.

```
* MEXICO SUBDUCTION MODEL
* CONTAINS A HIGH VELOCITY AREA BEHIND THE SIERRA
* MADRE DEL SUR
*
*-- RUN FLAGS AND PLOTTING SPECS.
*
MEXICO 500 SHOTS; UW4
3000100000
     0.000   605.000     0.040   100.000          5     0.000
     0.000    50.000     0.100    20.000          5     0.000
     0.000    10.000     0.500     1.000          1
      8.00      0.30      0.30      0.20       0.20
     70.00     5.000     0.000     0.000
*
*-- VELOCITY MODEL.
*
$OF.M84
*
*-- RAY SETS.
*
3000301100  MEXICO SUBDUCTION MODEL # 6
     0.000   100.000     0.800    0.0001   0.    0.    0.    0.
    281.00      0.00 -30.00000 -40.00000  05.    1.       0.00
    1    1
****
3000301100  MEXICO SUBDUCTION MODEL # 6
     0.000   100.000     0.800    0.0001   0.    0.    0.    0.
    281.00      0.00  30.00000  40.00000  05.    1.       0.00
    1    1
****
```



MEXICO 500 SHOTS, UW4
11-FEB-88 08:42:11

Interactive use of RAY84

RAY84 provides a degree of user interaction on VT100/Retrographics compatible graphics terminals. By interactively choosing and shooting ray sets, models may be tested and rayset angles determined much more rapidly than by using only the control file. To use RAY84 interactively, go to the first ray set in your control file and set IFLG(10) = 1 . The program will execute that ray set and then open communications for interaction by printing INSTRUCTIONS at the top, left of the screen. If you want to go directly to the interactive mode, also set IFLG(6) = 1 to delete the first ray set.

Interactive commands take two forms; variable-changing commands and general commands.

Variable Changing Commands:

Variables within the program may be changed by giving a command containing the variable name and '='. If a valid value for the variable follows the '=' sign, the variable will be reset to that value, otherwise you will be prompted for a new value. Following the prompt, if you enter a carriage return, the variable will be unchanged. The following list of changeable variables is grouped by function and default values are given.

Distance axis variables:

Range = 0.0, 60.0	Specify the minimum and maximum range for the distance axis.
Rmin = 0.0	Specify the minimum range for the distance axis.
Rmax = 60.0	Specify the maximum range for the distance axis.
Rsf = 0.25	Specify the range scale factor in plot units/km.
Dr = 10.0	Specify the distance between labeled tics on the distance axis in km.
Nsr = 10	Specify the number of intervals between labeled tics on the distance axis.
Rbias = 0.0	Specify the distance axis labeling offset in km. The minimum distance on the axis will be RMIN + RBIAS.
Rlth = 15.0	Specify the length of the distance axis. This re-calculates the value of RSF.

Depth axis variables:

Depth = 0.0, 40.0	Specify the minimum and maximum depth for the depth axis.
Zmin = 0.0	Specify the minimum depth for the depth axis.
Zmax = 40.0	Specify the maximum depth for the depth axis.
Zsf = 0.1	Specify the depth scale factor in plot units/km.
Dz = 10.0	Specify the distance between labeled tics on the depth axis in km.
Nsz = 5	Specify the number of intervals between labeled tics on the depth axis.
Zbias = 0.0	Specify the depth axis labeling offset in km. The minimum depth on the axis will be ZMIN + ZBIAS.
Zlth = 15.0	Specify the length of the depth axis. This re-calculates the value of ZSF.

Time axis variables:

Time = 0.0, 6.0	Specify the minimum and maximum time for the time axis.
Tmin = 0.0	Specify the minimum time for the time axis.
Tmax = 6.0	Specify the maximum time for the time axis.
Tsf = 1.0	Specify the time scale factor in plot units/sec.
Dt = 1.0	Specify the distance between labeled tics on the time axis in sec.
Nst = 10	Specify the number of intervals between labeled tics on the time axis.
Tlth = 6.0	Specify the length of the time axis. This re-calculates the value of TSF.

RV = 6.0	Reducing velocity (or NMO velocity).

Time variables:

Tstep = 0.1	Integration time step.
Tlimit = 99.	Time limit for calculation of each ray.
Tzero = 0.0	Start time for each ray.
Tprec = 0.0001	Precision of calculation.

Miscellaneous variables:

Tranct = 0.1	Velocity step for amplitude calculation.
Link = 0	Branch identifier for amplitude calculation.
Jtxt = <none>	Rayset ID string.
Gmax = 2.0	Maximum velocity gradient.

Symbol sizes:

Htc = 0.15	Size of tics and labels on the axes.
Hid = 0.15	Size of plot ID and axis labels. To eliminate axis labels, set HBS to a small positive number (0.00001).
Plotid = -none-	Plot ID to be plotted with the record section.

Observed data display:

Hobs = 0.1	Observed data symbol height.
Iobsym = 3	Observed travel-time data default symbol (0-31). Set IOBSYM = -1 to use QUAL from the travel-time file to define symbol.
Iobfrm = 0	Observed data file format. 0 - Standard format. 1 - TEKSEC standard format. 2 - Old standard format. 3 - amplitude file. 4 - RT11 format.

Calculated data display:

Hobs = 0.15 Calculated travel time symbol height.

Icline = 0 Calculated branch line type.

0 - Solid.	5 - 1/16" dash.
1 - 50 dots/inch.	6 - 1/8" dash.
2 - 25 dots/inch.	7 - 1/4" dash.
3 - 10 dots/inch.	8 - 1/2" dash.
4 - 5 dots/inch.	9 - 1" dash.

Pen Widths:

Newpen = 1 Sets pen width (1-6) for all lines in plot.

Pen = 1,1 Sets pen width (1-6) for individual parts of the plot (1-7). Syntax is Pen = 1,3 where the first entry is the plot characteristic and the second is the pen width.

Plot characteristics:

1 - Axes.
2 - Timing lines.
3 - Velocity rosette.
4 - Rays.
5 - Interfaces.
6 - Observed points.
7 - Calculated branch lines.

Velocity rosette variables:

Roser = 0.0	Distance for center of velocity rosette in km.
Roset = 7.0	Time for center of velocity rosette in sec.
Roses = 1.0	Size of arms of velocity rosette. Set ROSES to 0.0 or DISABLE ROSETTE for no rosette.
Rosign = 0.0	Direction of velocity rosette. 0.0 will plot the rosette both to the right and the left. 1.0 will plot to the right only. -1.0 will plot to the left only.

Timing line variables:

Tminl = -20.	Minimum time for timing lines.
Tmaxl = 20.	Maximum time for timing lines.
Dtline = 1.0	Time between timing lines. Set to 0.0 for no timing lines.

Isline = 2 Line type for timing lines.

0 - Solid.	5 - 1/16" dash.
1 - 50 dots/inch.	6 - 1/8" dash.
2 - 25 dots/inch.	7 - 1/4" dash.
3 - 10 dots/inch.	8 - 1/2" dash.
4 - 5 dots/inch.	9 - 1" dash.

General Commands:

To see a copy of the command set, enter the command,

HELP

After reading the HELP information, clear the screen of text with the command,

CLEAR

To trace a ray set, first specify the starting angle, ending angle, and number of rays by giving the command,

RAY

As you will be able to iteratively determine the proper ray angles, you can keep the number of rays shot in any trial to a minimum. I seldom use more than 15 . If you do not specify the number of rays, the previous value will be used.

To actually process the ray set, give the command,

PLOT

The interactive text will be cleared from the screen and rays will be calculated and plotted. After the completion of each ray, a report of its parameters will be listed on the screen. The tabulation on the screen will list for each ray, the starting angle, a status flag, the maximum interface encountered, the ray type, the range from the source, the total travel time, the absolute range with respect to the model axis, and the reduced travel time. The ray type is 0 for purely refracted rays; 1 for rays which have been reflected.

The status flag gives you the following information:

FLAG	Information
0.00	The time limit has been exceeded.
1.25	The ray code has been violated.
1.26-1.31	Failure to correctly calculate the intersection of a ray with an interface within the model.
1.50	Critical angle failure.
1.75	Ray requires more than 600 points.
1.80	Ray oscillates about an interface.
2.01	Ray left model at left edge.
2.02	Ray left model at right edge.
2.04	Ray left model at top.
2.08	Ray left model at bottom.
2.30-2.34	Ray ends at a requested interface (IFLG(7)).
2.40	Ray intercepts the target circle.
3.00	Successful ray guided by ray code.

To move the source position, give the command,

SOURCE

To change T0, TLIM, TSTEP, or TPREC, give the command,

TIME

or change them directly with the commands T0=, TLIM=, TSTEP=, or TPREC= .

To display travel times from an observed data file, give the command,

OBS

After a short time spent interactively shooting rays, the display will become too cluttered to see what you are doing. You can refresh the bare model by giving the command,

REPLOT

Many models are used which incorporate topography. When making final plots, it is desirable to place the label for zero depth at a point other than the top of the model. To do this, enter the command,

ZBIAS =

The program will request a value in km which will represent the new top of the model. After you REPLOT the screen, the depth axis will be appropriately labeled. ZBIAS affects only the axis labeling. Source depth is still defined relative to the top of the model. ZBIAS may also be defined in the control file.

To prevent the plotting of the date and time at the bottom left of the plot, give the command,

DISABLE DATE

The date and time may be restored with the command,

ENABLE DATE

Occasionally, you may want to halt the calculation of rays after a ray set has been initiated. To do this, suspend the operation of RAY84 by typing CNTRL/C . Then spawn the program RAYSTOP;

$ SPAWN/NOLOG PUB1:[REFRACT.LUETGERT.RAY83]RAYSTOP

Then restart RAY84 by typing,

$ CONTINUE

RAYSTOP creates a one line ASCII file (STOP.DAT) in your directory which tells RAY84 to stop what it is doing. If you use this feature often, you may want to establish an abbreviation for the above command. I use;

$ Q :== SPAWN/NOLOG PUB1:[REFRACT.LUETGERT.RAY83]RAYSTOP

After the ray set has been plotted, you can get a better look at the image by clearing the text from the screen using the command,

CLEAR

If you need another look at the ray set parameters, use the command,

REVIEW

to see those from the most recent ray set.

If the rays you are tracing require a ray code, first set IFLG(4) = 1 by typing,

ENABLE RAYCODE

Then, enter the number of entries in the ray code and the ray code itself after typing the command,

CODE

For more complex or long raycodes, you may use the alternate command,

PCODE

which accepts and interprets a compressed version of the ray code. As a simple example, where for CODE you might enter the ray code as 8,1,2,3,4,4,3,2,1 an equivalent entry using PCODE would be 1>4,4>1 .

To disable the ray code, set IFLG(4) = 0 by typing,

DISABLE RAYCODE

To erase the entire screen, enter the command,

ERASE

The entire screen will be erased.

Scaling parameters for the plot may be altered by the command,

SCALE

This command allows you to change the values of RMIN, RMAX, ZMIN,and ZMAX. This permits you to move about within a velocity model and concentrate on areas of interest. After changing scale factors, give the command REPLOT to reflect changes in the model plot. Alternatively, these values may be changed by the commands, RMIN=, RMAX=, ZMIN=, and ZMAX= .

There are a series of commands used in the automatic determination of individual travel time branches. For each of these, appropriate ray angles are determined and the last-defined number of rays, NRAY, are shot.

CRIT

This command is used in the determination of initial ray angles need to strike the critical point for a layer. The program will request an interface number. If you reply with a valid positive integer, the program will iteratively search for the critical refraction from that interface. If you reply with a 0 or ALL, the program will search for critical refractions from all interfaces. The program uses the last-defined starting and ending ray angles as initial values in the iterative search procedure.

REFLECT

This command is used in the determination of post-critical reflections. The program will request an interface number. If you reply with a valid positive integer, the program will search for and plot NRAY post-critical reflections from that interface. If you reply with 0 or ALL, the program will search for post-critical reflections from all interfaces.

REFRACT

This command is used in the determination of refracted arrivals. The program will request an interface number. If you reply with a valid positive integer, the program will search for and plot NRAY refracted arrivals from within the layer beneath that interface. If you reply with 0 or ALL, the program will search for refractions from all layers.

There are four commands which are used when modeling diffractions. For a complete explanation of these commands, see the later section on diffractions. The commands,

TARGET and NOTARGET

or

ENABLE TARGET and DISABLE TARGET

are used to enable and disable a target circle within the model which will terminate the propagation of rays. The commands,

DIFFRACT and NODIFFRACT

or

ENABLE DIFFRACT and DISABLE DIFFRACT

are used to place the program in the proper mode for modeling diffractions.

When setting up plots which are destined for the Versatec, you need to know the overall size of the plot being generated. This information is printed in the RAY.OUT file. You may also access this information interactively with the command,

SIZE

During the operation of the program, you may read a new model file by giving the command,

MODEL

The program will request the name of a model file and read it. Using this command and the REPLOT command allows you to run multiple models without exiting the program.

You may save ray set parameters by using the commands,

SAVE and SAVE CURRENT

The current values for a ray set will be saved in a file named SAV.OUT . This command allows you to find successful ray sets interactively and save them in a file for later use in constructing a final control file. SAVE stores the most recently shot set of rays; SAVE CURRENT stores the currently defined set of rays whether or not they have been shot.

DIRECT

This command is used in the determination of direct arrival raypaths. The program will search for and plot NRAY direct arrivals from the vertical raypath to the most distant direct arrival.

PRECRIT

This command is used in the determination of pre-critical reflections. The program will request an interface number. If you reply with a valid positive integer, the program will search for and plot NRAY pre-critical reflections from that interface. If you reply with 0 or ALL, the program will search for pre-critical reflections from all interfaces.

The critical angle search routine used by CRIT, REFLECT, REFRACT, DIRECT, and PRECRIT uses the last entered ray angles to start its search procedure. If you have been tracing rays to the right (positive angles), the critical points to the right of the source will be located; if you have been tracing rays to the left (negative angles), the critical points to the left of the source will be located.

There are two ways to exit the interactive mode of ray tracing. If you give the command,

RETURN

the program will return to processing ray sets from the control file beginning with the first ray set following the one which had IFLG(10) = 1 . If you give either of the commands,

EXIT or QUIT

the program will terminate.

Summary of Interactive Commands

HELP	List the command set.
CLEAR	Clears text from the screen.
REPLOT	Blanks the screen and redraws a blank model and axes. Requires two carriage returns (CR) after command.
SOURCE	Prompts the user for a new source location.
TIME	Allows the user to change any of the four fundamental timing constants T0, TLIM, TSTEP, or TPREC.
RAY	Prompts the user for ray angles ANGLE, ENDRAY, and RAYN for the next ray set to be shot.
PLOT	Shoots the next ray set.
REVIEW	Gives an abbreviated list of ray results for the previous ray set.
IFLG	Provides for resetting individual values of IFLG. IFLG(7) may be directly set to the interface number and sign normally set by the variable STFLG for stopping rays at intermediate interfaces. Note that if you wish to use the ray code, you must first set IFLG(4) = 0 .
CODE	Prompts the user to enter a ray code.
PCODE	Prompts the user to enter a ray code using abbreviated format.
OBS	Prompts the user for the name of an observed data file to be displayed.
SIZE	Reports the overall size of the plot in inches for use with VIEWR.
TARGET	Prompts the user for the range, depth, and radius of a target circle to shoot at to determine the starting time for diffracted rays.
NOTARGET	Resets the target flag so rays are not stopped by encountering a target circle.
DIFF	Prompts the user for parameters needed to simulate a diffraction and sets the diffraction flag, IFLG(9) . You must enter XPOSDF, XDIFF, ZDIFF, and TDIFF.
NODIFF	Resets the diffraction flag IFLG(9) .
RBIAS	Enter a bias for plotting the range axis. Following the command REPLOT, the labels on the range axis will be offset by rbias.
ZBIAS	Enter a bias for plotting the depth axis. Following the command REPLOT, the labels on the depth axis will be offset by zbias.

MODEL	Reread the model file. This is used when changing models in mid-stream or after editing.
SAVE	Save the last executed ray set in SAV.OUT .
SAVE CURRENT	Save the last defined ray set in SAV.OUT .
ERASE	Erases the entire screen.
SCALE	Allows changing the values of RMIN, RMAX, RSF and ZMIN, ZMAX, ZSF.
CRIT	Searches for critical angles.
REFLECT	Searches for and plots post-critical reflections.
REFRACT	Searches for and plots refractions.
DIRECT	Searches for and plots direct arrivals.
PRECRIT	Searches for and plots pre-critical reflections.
AUTO	Searches for and plots refractions and post-critical reflections from all interfaces.
RETURN	Returns control to the control file starting at the ray set following that which had IFLG(10) set to 1 .
EXIT / QUIT	Terminate the program. Requires two (CR) after command.
ENABLE NMO	Observed and calculated travel times are plotted with a hyperbolic normal moveout correction using the reducing velocity, RV, as the correction velocity. $T = \sqrt{T^2 - X^2/RV^2}$
DISABLE NMO	Observed and calculated travel times are plotted with a reduced travel time using the reducing velocity, RV. $T = T - X/RV$
ENABLE RAYCODE	Rays will be guided by the current ray code.
DISABLE RAYCODE	The ray code will not be used.
ENABLE DATE	The date and time will be plotted at the bottom left of the plot.
DISABLE DATE	The date and time will not be plotted.

Output from RAY84

When Ray84 is run, the program will request the names for up to three output files. The default names are RAY.OUT, RAY.SUM, and RAY.AMP .

RAY.OUT - This is the main output file from the program. It contains information on the interfaces, velocity distribution, and ray sets. The amount of detail in this file depends on the values of JFLG(1) and IFLG(5) chosen.

RAY.SUM - This file contains the ray summary. It is information already contained in RAY.OUT . RAY.SUM contains only the computed travel times and ranges. JFLG(4) determines if anything is written to this file.

RAY.AMP - This file contains the ranges, travel times, and complex amplitudes for each ray. This file is modified by the user and then used as input to R83PLT . JFLG(7) determines if anything is written to this file.

SAV.OUT - This file is created by using the command SAVE and contains the saved values for ray sets.

Graphical output from RAY84

The graphical output of RAY84 is controlled by a combination of JFLG(3), JFLG(10) and the optional interactive prompts generated by the plot library. JFLG(3) governs the combination of model and travel time plot to be plotted.

If JFLG(10) = 2 or 3, the normal plotting library prompts are given, giving you a choice of output to the terminal, to a BATCH.PLT file for later plotting, both, or neither.

If you set JFLG(10) = 0 or 1, the output will go only to the VT100 terminal and the appropriate screen size will be determined for you. You may find the size used by looking at the RAY.OUT file or using the interactive command SIZE. If JFLG(10) = 1, the image you see will be to the proportions you specified in your control file, but reduced to fit within the bounds of the screen. If JFLG(10) = 0, the three scaling factors will be recomputed to produce a plot which optimally fills the screen with a SIZE = 8.0 (default value for the plotting library).

Plotting Observed Travel Time Data

The program will ask for the name of this file only if JFLG(6) = 1 . Observed travel time data are found in ASCII files (pick files) under one of five recognized formats. The appropriate format is selected by the variable IOBFRM which has a default value of 0. Under any of the formats, a line with * in the first column is considered to be a comment line and is ignored. The formats are:

0 Default standard format

The first line is a global parameter line, FORMAT (4F10.0, I5), containing RNG, VR, TADD, FUDGE, and ICHAR. RNG is the distance in kilometers of the source with respect to the model. VR is the reducing velocity which has been applied to the subsequent tabulated pick times. TADD is an additive constant to be applied to all times. FUDGE is the range scaling fudge-factor. R=R*FUDGE. If FUDGE = 0.0 on input, it is set to 1.0 . ICHAR is the character code to be used for plotting observed travel times (0-14) .

Travel time data lines, FORMAT (A4, 3F10.0, I3), contain ID, DELT, TIME, AZIM, and QUAL.

ID is a four character identification, usually the location number. It is ignored by this program. DELT is the distance from the shot to the receiver. TIME is the travel time reduced by VR. AZIM is the azimuth of the receiver from the shot in degrees clockwise from north. QUAL is a number which may be used to identify an arrival as a member of a class of arrivals, *i.e.* quality of a pick or branch number. QUAL may be used to select the plotting character; see IOBSYM below.

Control lines; NEW, END, CHAR

If a line is read with NEW in the first three columns, the program expects that the next data line will contain global parameters. This allows you to change global parameters within a file. This is particularly useful for combining pick files which have been picked with different reducing velocities.

If a line is read with END in the first three columns, that line is treated as an end-of-file and no further data are read.

The plotting character may be changed by a line containing CHA in the first three columns followed by a blank and a numeric value. *i.e.* CHA 3 or CHARACTER 4.

1 TEKSEC format

This format contains only travel time data. Global parameters are assumed. VR is assumed to be the same as the reducing velocity being used for the record section plot. TADD is assumed zero. Travel time data lines are the same as for Format 0 above. Control lines, as defined in Format 0, are recognized although they are non-standard for TEKSEC format.

2 Old standard format

This format is identical to Format 0 except that there is no azimuth entry in the travel time data lines. The travel time data lines, FORMAT(A4, 2F10.0, I3), contain ID, DELT, TIME, and QUAL.

3 Amplitude file format

This format is that written to RAY.AMP by RAY84.

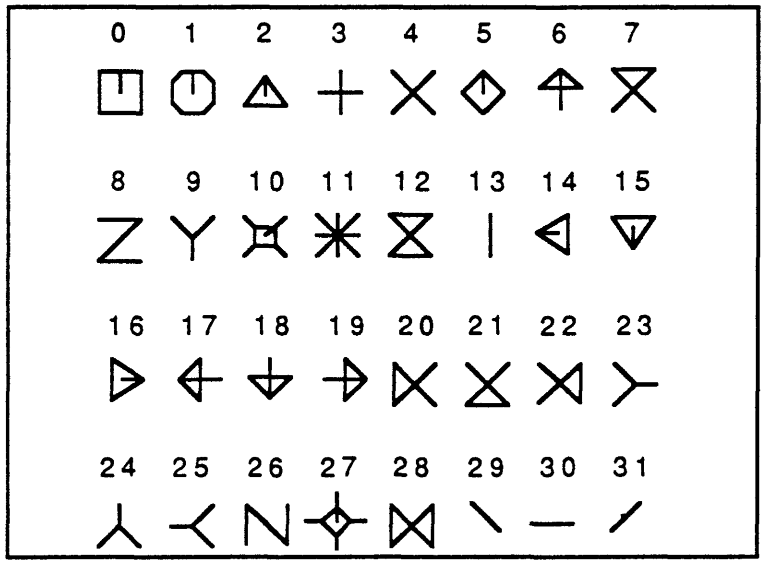
4 RT11 format

This is the pick file format produced by the digitizing computer. Like TEKSEC format, global parameters are assumed.

Travel time data lines, FORMAT (4X, 2I4, 7X, F7.0, 2F8.0), contain ID, QUAL, TIME, DELT, and AZIM.

Specifying the symbol to be used for plotting.

The size of the plotted symbol is given by HOBS. [default value = .15] The plotted symbol is determined by IOBSYM. [default value = 3 (+)]
For IOBSYM = 0-31, use the appropriate index symbol from the table below.



For IOBSYM < 0, use the value of QUAL from each travel time data line to determine the symbol.

If IOBFRM = 0,1, or 2, the symbol may be changed by a CHAR control line.

If IOBFRM = 3 (amplitude file) and IOBSYM = 99, refracted arrivals are plotted as X; reflected arrivals as +.

Modeling Diffractions with RAY84

Arrival times for diffracted arrivals are approximated by first calculating the portion of the ray between the source and the diffraction point. Then, the diffraction point is used as a new source. The travel time and X-distance between the true source and the diffraction point are added to each ray calculated from the diffraction point.

To find the travel time between the source and the diffraction point, use the interactive command TARGET to specify the range, depth, and radius of a target circle within the model. If the target parameters have already been specified, the target circle may be enabled by the command ENABLE TARGET. All further rays encountering the target circle will be stopped and the time reported will be the travel time from the source to the target circle. When setting the radius of the target circle, remember that the target criterion is met only if the end of a ray segment is within the target circle; a ray with time steps so large that a ray segment intercepts the target circle without having one of its end points within the circle will not be stopped. To turn off the target circle feature, use the command NOTARGET or DISABLE TARGET.

Once the travel time to the diffraction point has been determined, the diffracted portion of the rays can be calculated. If you are in interactive mode,

1) Turn off the target circle by issuing the command NOTARGET or DISABLE TARGET.

2) Give the command DIFF . The program will ask you to supply the X-position of the true source (XPOSDF), the range and depth of the diffracting point (XDIFF, ZDIFF) and the travel time to the diffraction point from the source (TDIFF). This also sets a flag in the program (IFLG(9)) to add XPOSDF-XDIFF and TDIFF to the calculated ranges and times for diffracted rays. If the diffraction point parameters have been entered, IFLG(9) may be set by using the command ENABLE DIFFRACTION.

3) Give the command RAY to set the angles of emergence from the diffraction point and the number of rays.

4) Give the command PLOT to plot the diffracted rays.

5) When you are done, reset the diffraction flag by giving the command NODIFF or DISABLE DIFFRACTION.

If you want to calculate and plot diffracted rays non-interactively, set up a ray set in your command file, making sure you;

1) set IFLG(9) = 1 on the first card.

2) set T0 = travel time from source to diffracting point (first entry on the second card), and

3) set XPOS and ZPOS to the coordinates of the diffraction point and XPOSDF to the X-location of the true source on card 3.

Remember that strictly speaking, diffractions are not a ray phenomenon and although travel times may be correctly calculated, these rays should not be used when making amplitude calculations.

Running RAY84 in Batch Mode

If you are not using RAY84 in its interactive mode, it is desirable (and courteous to other users) to run the program in Batch mode. This is especially useful for doing final runs in which you are generating amplitude files or if you are extensively using the automatic branch finding features.

To submit a batch RAY84 run you must create a batch control file as in the following example in which the batch control file is named TESTBAT.BAT, the RAY84 control file is named TESTBAT.R84 and the observed data file is named TESTBAT.OBS .

TESTBAT.R84 and TESTBAT.OBS need not be changed to use them in batch mode rather than from the terminal. The interactive features of RAY84 invoked by IFLG(10) in the ray sets are automatically defeated in batch mode. The prompting information required to start the program is no different than when running the program from a terminal. Thus, it is important to be careful in constructing the TESTBAT.BAT file so that it contains the proper set of responses.

The following batch control file assumes that you have requested an amplitude file (JFLG(7)) and will be displaying an observed data file.

```
$ASSIGN CRUD SYS$PRINT
$CWD SUBDIR
$RAY84
TESTBAT.R84
TESTBAT.AMP
TESTBAT.OBS
2
1
```

A line-by-line explanation:

$ASSIGN CRUD SYS$PRINT

The batch job is run from your home directory. Output from the batch process is normally directed to SYS$PRINT which is connected to the printer by default. We defeat this by assigning SYS$PRINT to a null file (CRUD in this case). The batch processor deals with this by creating a .LOG file in your home directory. In this example, the file would be named TESTBAT.LOG .

$CWD SUBDIR
If your control file and observed data file are not in your home directory, you will need this line to change the working directory.

$RAY84
This line invokes RAY84. Rigorously, this is an abbreviation for
$ RUN PUB1:[REFRACT.LUETGERT.RAY83]RAY84.EXE

TESTBAT.R84
The name of the control file.

TESTBAT.AMP
The name of the amplitude file to be created.

TESTBAT.OBS
The name of the observed data file. If JFLG(6) is equal to zero in your control file, do not include this line.

2
This is a response to the plot library prompts. A 2 tells the plotting routine to do all plotting to a BATCH.PLT file only. If you wish no plot at all enter a 4 on this line and remove the next line.

1
Directs the plot output to a BATCH.PLT file.

The above batch job is submitted to the batch queue by giving the following command from a terminal.

$SUBMIT TEST.BAT

If the run time is likely to be small, you can use the Quick Batch queue via the command,

$SUBMIT/QUEUE=QUICK TESTBAT.BAT

To monitor the status of your batch job, issue the command,

$SHOW QUEUE/BAT

R83PLT

R83PLT is used to calculate and plot Ray Theoretical Synthetic Seismograms (McMechan and Mooney, 1980) using the output from RAY84. When JFLG(7) is set ≠ 0 in the input file to RAY84, an amplitude file is created containing dummy control variables in the format needed for R83PLT (described below) and a series of lines describing the distance, time, complex amplitude and branch of each ray calculated. The parameters that follow are found at the beginning of the RAY.AMP file generated by RAY84. You may need to change some of these parameters before plotting the synthetics. When this step is complete R83PLT can be run. Some of the parameters listed below are deduced by RAY84 and written to RAY.AMP . Following these parameters are the tabulated complex amplitudes used to construct synthetic seismograms. Each line contains the model range, travel time, complex amplitude and LINK number for a ray. For each distance at which a synthetic seismogram is to be plotted, the list is scanned for adjacent rays having the same LINK number which bracket the requested distance in range. These values are used to interpolate a contribution to the synthetic seismogram.

This section of the file may be preceded by any number of comment lines containing an asterisk (*) in the first column. The information in this section controls the program's output to files and the appearance of the plotted output.

Line 1 - Title (80A1).
The title is printed in the run header of the output list and appears on the plot.

Line 2 - JFLG (10I1) Flags to be in effect for the entire run.

JFLG(1) - Print flag for input data.
=0 No list output.
=1 List output to R83PLT.OUT .

JFLG(2) =0 Not currently in use.

JFLG(3) - Instrument response correction.
=0 No instrument response correction.
=1 Prompt for instrument correction.
=2 Correct for instrument response.

JFLG(4) - Trace spacing.
- =0 Uniform spacing of synthetic traces.
- =1 Ranges for traces specified on line(s) 12.

JFLG(5) - Measurement units.
- =0 Scaling factors defined in terms of inches.
- =1 Scaling factors defined in terms of cm.

JFLG(6) - Not currently in use.

JFLG(7) - Not currently in use.

JFLG(8) - Not currently in use.

JFLG(9) - Not currently in use.

JFLG(10)- Display and plot output.
- =0 Assume that plotted output will be to the VT100 only and skip the initial plot package prompts. The scaling factors for the axes specified on lines 3-5 will be ignored and the lengths of the axes will be independently re-computed to fill the screen.
- =1 Assume that plotted output will be to the VT100 only and skip the initial plot package prompts. The plot will be increased or decreased in overall size to fit the screen.
- =2 Use normal plotting prompts. The plot will be increased or decreased in overall size to fit the VT100 screen. If a BATCH.PLT vector file is created, it will have the size specified by lines 3-5 below. See the list output file RAY.OUT for the size specifications required to run VIEWR.
- =3 Use normal plotting prompts. The scaling factors for the axes specified on lines 3-5 will be ignored and the lengths of the axes will be independently re-computed to fill the screen. If a BATCH.PLT vector file is created, it will be small enough to fit on a VERSATEC sheet without scaling.

When you are first starting to plot a synthetic section you will probably want to set the JFLGs to 1000000002.

Lines 3 and 4 define the distance and time axes for the travel time plot. The values given for RMIN and RMAX need not correspond to the bounds of the defined velocity model.

Line 3 - RMIN, RMAX, RSF, DR, NSR, RBIAS (4F10.0, I10, F10.0)
Range axis plot specifications.

RMIN = Minimum range (kilometers).

RMAX = Maximum range (kilometers).

RSF = Range scale factor (in/km or cm/km).

DR = Distance between labeled tics (kilometers).

NSR = Number of small divisions between labeled tics.

RBIAS = Bias to be applied to the range axis labeling. This may be used to make the range axis reflect distance from a shot point. This variable only affects the axis labeling. Subsequent references to locations within the model, such as position of the source, are given in absolute model coordinates.

Line 4 - TMIN, TMAX, TSF, DT, NST (4F10.0, I10)
Time axis plot specifications.

TMIN = Minimum time (seconds).

TMAX = Maximum time (seconds).

TSF = Time scale factor (in/sec or cm/sec).

DT = Distance between labeled tics (seconds).

NST = Number of small divisions between labeled tics.

Line 6 - RV, HTC, HID (3F10.0)
Plot specifications.

RV = Reducing velocity in km/sec (use 0.0 or 10000.0 for an unreduced plot).

HTC = Size of tics and axis labels in inches or cm. An entry of 0.0 sets HTC to a default size of 0.3. A negative entry sets HTC to zero and eliminates tics and axis labels.

HID = Size of the plot ID in inches or cm. An entry of 0.0 sets HID to a default size of 0.3. A negative entry sets HID to zero and eliminates the plot ID.

Line 6 - ROSER, ROSET, ROSES, ROSIGN (4F10.0)
Velocity rosette specifications.

ROSER = Range of center of velocity rosette (kilometers).

ROSET = Time for center of velocity rosette (seconds).

ROSES = Length of arms of velocity rosette in inches or cm. Set equal to zero for no rosette.

ROSIGN = Direction of the velocity rosette. +1.0=right; -1.0=left; 0.0=both.

Line 7 - X0, XF, DELX, DTS, TMINS, XSHIFT (6F10.0)
Seismogram plotting information.

X0 = Range of first seismogram (kilometers).

XF = Range of last seismogram (kilometers).

DELX = Range step between seismograms. The number of seismograms will equal (XF-X0)/DELX .

DTS = Time step of the seismograms (seconds).

TMINS = Minimum time of the seismogram (seconds).

XSHIFT = Distance in km that the source is to be shifted laterally relative to the origin of the coordinate system (*i.e.* XPOS from RAY84) .

Line 8 - SF, AMPL, CLIP, POWER, IFSF2, IFCLIP, NSMOOTH (4F10.0, 3I5)
Parameters governing seismogram traces.

SF = Scale factor for the seismogram amplitudes. Used for range-scaled traces.

AMPL = Scale factor for the seismogram amplitudes. Used for normalized traces.

CLIP = Amplitude at which clipping is to be done (in units of DELX).

POWER = Scale factor for distance dependent amplitude scaling.

IFSF2 =0 Traces self-scaled with amplitude AMPL.
=1 No distance scaling.
=2 Seismogram amplitude will be scaled by the square root of the range.
=3 Scaled by X.
=4 Scaled by X^2.
=5 Scaled by $X^{1.5}$.
=6 Scaled by X^{POWER}.

IFCLIP =0 No clipping.
=1 Extreme amplitudes are to be clipped.

NSMOOTH = Number of times a triangular smoothing wavelet is to be applied.

Line 9 - NSRCE, IFCONV (2I5)
Information concerning the source function.

NSRCE = Number of samples in the source function.

IFCONV =0 No convolution of source function.
=1 The source function is to be convolved with the impulse response.

Note that when NSRCE is 0, IFCONV must also be 0 .

Line(s) 10 - SRCE(I), (I=1,NSRCE) (10F8.0)
Used only when NSRCE > 0 .

SRCE(I) = List of points defining the source pulse, time step should be DTS (defined on line 7) .

Line 11 - NRNG (I5)

NRNG = Number of specified ranges (given on next line(s)) at which to plot synthetic traces. Used only if JFLG(4) = 1 . If JFLG(4) = 0, line 11 and line(s) 12 should not appear in the file.

Line(s) 12 - RSEIS(I), I=1,NRNG (10F8.0)

RSEIS = Ranges at which to calculate and plot synthetic traces.

Lines 13 - X(I), T(I), A_R(I), A_I(I), LINK(I) (4E15.7, I10)

These are the amplitudes output by RAY84.

References:

Cerveny, V., I.A. Molotkov, and I. Psencik, 1977. *Ray Method in Seismology*, 214 pp., Charles University Press, Prague.

McMechan, G.A. and W.D. Mooney, 1980. Asymptotic ray theory and synthetic seismograms for laterally varying structures: theory and application to Imperial Valley, California, *Bull. Seism. Soc. Am.* **70**, 2021-2035.