Tomography Program: tomo_sp_cu_s, v1.1
(Menu and Input-Output Description)

tomo_sp_cu_s is a program designed for inversion of regional or global scale surface wave group or phase velocity measurements to estimate 2-D models of the distribution and strength of isotropic and azimuthally anisotropic velocity variations. This program includes only Gaussian beams method. This manual describes program Menu and formats of the input-output files.
/

1 Command line.

tomo_sp_cu_s ***input_data_file*** ***output_prefix*** ***period***

The following invocation operands are supported by **tomo_sp_cu_s** program:

input_data_file - data file within observations;
output_prefix - part of the output files name. Full file name is a catenation of output_prefix, underscore character **"_"** , period, underscore and suffix, e.g., ***output_prefix_period_suffix.*** Possible values for suffixes are: **".1", ".1_%_", ".res",** etc. See for details in section 4.
For example, if we have the following call:
…> tomo_sp_cu_s clean_data_R_100 TEST_R_200_100 50
then output file names will be: **TEST_R_200_100_50.1 , TEST_R_200_100_50.1_%_ , TEST_R_200_100_50.res, …;**
period - observed period, integer number, (s).

2 Commands.

2.1 General tomo_sp_cu_s commands.

When started, **tomo_sp_cu_s** program reads from ***stdin*** the following commands:

h[elp] or **?** - print help menu to the screen;
me[nu] - enter settings menu;
v[iew] - view current settings;
def[aults] - reset current settings to defaults;
go - run program;
q[uit] or **ex[it]** - quit program.

2.2 Menu commands.

When entered general command **me[nu], tomo_sp_cu_s** program reads from ***stdin*** the following Menu commands:

r or..**q** or..**x** - return from Menu to the main program;
v[iew] - view current settings;
0,1,2,...,29 - upgrade settings, for more details see sections **2.3** and **2.4**.

2.3 Default Settings.

The following text is a screen output for default setting (Menu output) of the global or Menu **v[iew]** command. Usually this output represents current parameters settings. You can get default settings, if enter **v[iew]** command immediately after program start or after **def[aults]**

command. Each line of the screen output consists of three parts: command name (number), the sense of commands and command parameter(s).

```
DATA  CHARACTERISTICS:
 0.) Model are given?.................(toggle)...         0
 1.) Weights are given?...............(toggle)...         0
 2.) Number of paths < ..........................     100000
 3.) Selection of paths ..............(toggle)...         0
PLOT CHARACTERISTICS:
 4.) Path density & azim. coverage ?...(toggle)...         0
 5.) Limits of the map (latitudes,step) ............    -20.00   89.00    1.00
 6.) Limits of the map (longitudes,step) ...........     0.00  359.00    1.00
 7.) Map of deviations in %?...........(toggle)...         1
 8.) Rejecting too strange data?.......(toggle)...         0
 9.) Rejecting data by wavelength?.....(toggle)...         0
10.) Step,X-zone,R/L,G/P,Size of cells:iso,anis...    0.500 2.000 R G  2.500 7.000
11.) anisotropy: 0 - no, 1 - 2psi, 2 - 4psi.......         0
12.) 0PSI: alpha1, alpha2, sigma1, sigma2...........  1000.000  1.000  250.000  250.000
13.) 2PSI: alpha1,        sigma1, sigma2...........   800.000        500.000  500.000
14.) 4PSI: alpha1,        sigma1, sigma2...........  1200.000        500.000  500.000
15.) 0PSI-prec, 2PSI-prec, 4PSI-prec ..............      8.0    8.0    8.0
16.) Contour file name.............................  contour.ctr
17.) Model file name...............................  model_map.ctr
18.) Model name file for slowness determination ....  PREM.MODEL
19.) Output residuals?.................(toggle)...         0
20.) Covariance matrix?................(toggle)...         0
21.) Apply geogr --> geoc for input....(toggle)...         1
22.) Apply geoc --> geogr for output...(toggle)...         1
23.) Anisotropy pole: lat, lon.....................     0.00   0.00
25.) Produce resolution analysis map?..(toggle)...         0
26.) Use new input data format R1/R2?..(toggle)...         0
27.) Use RAY TRACER mode?..............(toggle)...         0
28.) Make response map for some points?(toggle)...         0
29.) Path density wts for iso and aniso parts..... -0.1471300    -0.1471300
```

2.4 Menu command and Menu output description.

As mentioned before the part of Menu command names are just numbers in range **0** ÷ **29**. Those commands may be entered from the screen (screen dialog), from file or shell script. For all cases **tomo_sp_cu_s** program uses ***stdin*** device for input data and ***stdout*** device for output data.
There are three types of commands - toggle, parametric and mixed.
Toggle command is an answer to the menu question - "**yes**" (1) or "**no**" (0). To enter such a command just enter corresponding command number, if current state was (0), it becomes (1), if was (1), it becomes (0). **`v[iew]`** command shows us the current setting (0 or 1).
Parametric subcommands suppose some kind of ***stdin/stdout*** dialog. Usually program asks to input some parameter(s). See below more detail about this staff.
Mixed command toggles current status as a toggle command but also makes request for some parameters if you toggle the current status from (0) to (1), only. If the current status is (1) and mixed command is entered, it doesn't request any parameters.

Command 0 - Define reference model (toggle).

Menu line:

```
0.) Model are given?.................(toggle)...          0
                                                ↑
                                        toggle parameter
```

Description: ***toggle parameter*** = 0 - reference model is a constant;
toggle parameter = 1 - reference model file must exists in working directory and have the name "model.map", otherwise name must be strictly defined by command **17**. Path to file may be absolute or relative to working directory.

Command 1 - Setup weights (toggle).

Menu line:

```
1.) Weights are given?...............(toggle)...          0
                                                ↑
                                        toggle parameter
```

Description: ***toggle parameter*** = 0, no weights;
toggle parameter = 1 - weights are given.

Command 2 - Setup maximum number of lines for input data file (parametric).

Menu line:

```
2.) Number of paths < ..........................      70000
                                                ↑
                                              ipath
```

Description: ***ipath*** defines maximal number of lines from the beginning of input data file which program will be read. Default value: 70000.

Dialog:
…> enter maximum number of paths:
…> ***ipath***

Command 3 - Select paths by rectangular border for rays (mixed).

Menu line:

```
3.) Selection of paths ..............(toggle)...          0
                                                ↑
                                        toggle parameter
```

Description: ***toggle parameter*** = 0, don't use selection, don't use *Dialog*;
toggle parameter = 1 - select paths, use *Dialog* to define border.

Dialog:
…> Borders for rays are:
…> Latitudes: -20.00 89.00
…> Longitudes: 0.00 359.00
…> Any changes?(Y/N)
…> ***answer***
…> ***rlat0 rlat1 rlon0 rlon1***

Where,
rlat0,rlat1 - range by latitude (°), ***rlat0 <= rlat1;***

rlon0,rlon1 - range by longitude (°), ***rlon0 <= rlon1.***
If ***answer*** is ***Y*** or ***y*** additional line required, otherwise end of dialog.

Command **4 -** Output density and coverage maps (toggle).
*Menu line**:***

```
4.) Path density & azim. coverage ?...(toggle)...          0
                                                           ↑
                                                  toggle parameter
```

Description: ***toggle parameter*** = 0, don‘t output maps;
toggle parameter = 1 - output maps in files: **xxx.res, xxx.azi**.

Command **5 -** Setup latitude grid boundaries and step for all output maps (parametric).
*Menu line**:***

```
5.) Limits of the map (latitudes,step) .............    -20.00   89.00    1.00
                                                         ↑        ↑        ↑
                                                       lat0     lat1   lat_step
```

Description: ***lat0*** - lower grid boundary, (°);
lat1 - upper grid boundary, (°), ***lat1 > lat0***;
lat_step - grid step (increment), (°), ***lat_step >= 1***
Parameters ***lat0,lat1,lat_step*** must be integer.
Dialog: …> enter limits for latitudes and increment
…> ***lat0 lat1 lat_step***

Command **6 -** Setup longitude grid boundaries and step for all output maps (parametric).
*Menu line**:***

```
6.) Limits of the map (longitudes,step) .............     0.00  359.00    1.00
                                                         ↑       ↑       ↑
                                                       lon0    lon1   lon_step
```

Description: ***lon0*** - lower grid boundary, (°);
lon1 - upper grid boundary, (°), ***lon1 > lon0***;
lon_step - grid step (increment), (°), ***lon_step >= 1;***
Parameters ***lon0,lon1,lon_step*** must be integer.
Dialog: …> enter limits for longitudes and increment
…> ***lon0 lon1 lon_step***

Command **7 -** Output isotropic velocity map in percent (toggle).
*Menu line**:***

```
7.) Map of deviations in %?............(toggle)...          1
                                                            ↑
                                                   toggle parameter
```

Description: ***toggle parameter*** = 0, don‘t output map;
toggle parameter = 1 - output maps in files: **xxx.1_%_**

Command 8 - Reject paths by velocity threshold (mixed).

*Menu line**:***

```
8.) Rejecting too strange data?.......(toggle)...          0
                                                  ↑
                                          toggle parameter
```

Description: ***toggle parameter*** = 0, don't use selection, don't use *Dialog*;
toggle parameter = 1 - select rejection, use *Dialog* to define threshold.

Dialog: …> `enter threshold in % for rejection`
…> ***`threshold`***

Comments: If ***`|(v(i) -v)/v|*100>=threshold,`*** the measurement with velocity ***`v(i)`*** is rejected. ***`v`*** is the average velocity for all input data.

Command 9 - Reject paths by number of wavelength (mixed).

*Menu line**:***

```
9.) Rejecting data by wavelength?.....(toggle)...          0
                                                  ↑
                                          toggle parameter
```

Description: ***toggle parameter*** = 0, don't use selection, don't use *Dialog*;
toggle parameter = 1 - select rejection, use *Dialog* to define number of wavelengths λ .

Dialog: …> `enter number of wavelength for rejection`
…> ***`numw`***

Comments: Number of wavelength, ***`numw`*** must be integer positive number, usually, =5.
To compute wavelength program uses a phase velocity for given period defined by 1-D model (command #**18**). Type of wave must be setup by command #**10**, fileld ***`t_wave`***. Data with epicentral distance less than ***`numw`*** * λ will be rejected.

Command 10 - step of integration and others (parametric).

*Menu line**:***

```
10.) Step,R/L,G/P, Size of cells:iso,anis...  0.500  R   G  2.500 7.000
                                               ↑    ↑   ↑    ↑  ↑
                                            step  t_wave t_vel si  sa
```

Description: ***`step`*** - step of integration, (°), for Gaussian method;
`t_wave` - type of surface wave, **R** - Rayleigh, **L** - Love;
`t_vel` - type of velocity, **G** - group, **P** - phase;
`si` - size of isotropic cell, (°);
`sa` - size of anisotropy cell, (°).

Dialog: …> `Step of integration`
…> ***`step`***
…> `enter type of wave (Rayleigh, Love): R or L`
…> ***`t_wave`***

```
…> enter type of velocity (Group, Phase): G or P
…> t_vel
…> enter size of main cell (degree)
…> si
…> enter size of anisotpopy cell (degree)
…> sa
```

Command 11 - Setup type of topography (parametric).

Menu line:

```
11.) anisotropy: 0 - no, 1 - 2psi, 2 - 4psi.....          0
                                                          ↑
                                                        aniz
```

Description: ***aniz*** - =0 - purely isotropic tomography, =1 - 2Ψ anisotropic tomography, =2 - 2&4Ψ anisotropic tomography.

Dialog:

```
…> enter type of anisotropy: 0 - no, 1 - 2psi, 2 - 2&4psi
…> aniz
```

Command 12 - Setup regularization parameters for isotropic tomography (parametric).

Menu line:

```
12.) 0PSI: alpha1, alpha2, sigma1, sigma2..........  1000.000  1.000  250.000  250.000
                                                     ↑        ↑      ↑        ↑
                                                   alpha0   damp   sigma0   sigma0x
```

Description:

alpha0 - smoothing damping coefficient;

damp - path density damping, range - 0.0 ÷ ∞, = 0.0 - no damping, =1 - regular value, >1 - more strong damping that the regular one;

sigma0 - Gaussian σ for smoothing, (km);

sigma0x - not used in program, but must setup to some value in Dialog, reserved for the future extensions.

Dialog:

```
…> enter regularization parameters for 0PSI
…> alpha0 damp sigma0 sigma0x
```

Command 13 - Setup regularization parameters for 2Ψ anisotropic tomography (parametric).

Menu line:

```
13.) 2PSI: alpha1,       sigma1, sigma2...........   800.000        500.000  500.000
                                                     ↑              ↑        ↑
                                                   alpha1         sigma1   sigma1x
```

Description:

alpha1 - smoothing damping coefficient;

sigma1 - Gaussian σ for smoothing, (km);

sigma1x - not used in program, but must setup to some value in Dialog, reserved for the future extensions.

Dialog:

```
…> enter regularization parameters for 2PSI
…> alpha1 sigma1 sigma1x
```

Command **14 -** Setup regularization parameters for 4Ψ anisotropic tomography (parametric).

*Menu line**:***

```
14.) 4PSI: alpha1,       sigma1, sigma2.....  1200.000        500.000  500.000
                                              ↑              ↑       ↑
                                           alpha2         sigma2  sigma2x
```

Description: ***alpha2*** - smoothing damping coefficient;
sigma2 - Gaussian σ for smoothing, (km);
sigma2x - not used in program, but must setup to some value in *Dialog*, reserved for the future extensions.

Dialog: …> `enter regularization parameters for 2&4PSI`
…> ***alpha2 sigma2 sigma2x***

Command **15 -** Setup cutting value for smoothing Gaussian accuracy (parametric).

*Menu line**:***

```
15.) 0PSI-prec, 2PSI-prec, 4PSI-prec ..............      8.0    8.0    8.0
                                                       ↑      ↑      ↑
                                                    prec0  prec1  prec2
```

Description: ***prec0*** - cutting value for 0Ψ ;
prec1 - cutting value for 2Ψ ;
prec2 - cutting value for 4Ψ .

Dialog: …> `enter accuracy for 0PSI, 2PSI, 2&4PSI`
…> ***prec0 prec1 prec2***

Command **16 -** Define path to `contour` file (parametric).

*Menu line**:***

```
16.) Contour file name.............................  contour.ctr
                                                     ↑
                                                  ctr_f_name
```

Description: ***ctr_f_name***- absolute or relative to working directory path name to file with contour. By default file must be in working directory and must have name `contour.ctr`.

Dialog: …> `enter CONTOUR file name`
…> ***ctr_f_name***

Command **17 -** Define path to reference model file (parametric).

*Menu line**:***

```
17.) Model file name...............................  model_map.ctr
                                                     ↑
                                                  mod_f_name
```

Description: ***mod_f_name***- absolute or relative (to working directory) path name to file with reference model. By default file must be in working

directory and must have name `model_map_ctr`.

Dialog: …> `enter MODEL file name`
…> ***`mod_f_name`***

Command 18 - Define path to 1-D model for rejection data by wavelength (**command 9**).
*Menu line**:***
`18.) Model name file for slowness determination ....  PREM.MODEL`
↑
`1D_f_name`

Description: ***`1D_f_name`*** - absolute or relative (to working directory) path name to file with 1-D model. By default file must be in working directory and must have name `PREM.MODEL`.

Dialog: …> `enter MODEL file name for slowness determination`
…> ***`1D_f_name`***

Command 19 - Output initial and final residuals (toggle).
*Menu line**:***
`19.) Output residuals?.................(toggle)...        0`
↑
`toggle parameter`

Description: ***toggle parameter*** = 0, don‘t output residuals;
toggle parameter = 1 - output residuals in file: **xxx.resid**

Command 20 - Output diagonal of covariance matrix (toggle).
*Menu line**:***
`20.) Covariance matrix?................(toggle)...        0`
↑
`toggle parameter`

Description: ***toggle parameter*** = 0, don‘t output matrix diagonal;
toggle parameter = 1 - output matrix diagonal in file: **xxx.dr**

Command 21 - Apply coordinate transformation to input data from geographical coordinate system to geocentrical (toggle).
*Menu line**:***
`21.) Apply geogr --> geoc for input....(toggle)...         1`
↑
`toggle parameter`

Description: ***toggle parameter*** = 0, don‘t apply transformation;
toggle parameter = 1 - apply.

Command 22 - Apply coordinate transformation to output data from geocentrical

coordinate system to geographical (toggle).

Menu line:

```
22.) Apply geoc --> geogr for output...(toggle)...          1
                                                            ↑
                                                  toggle parameter
```

Description: ***toggle parameter*** = 0, don‘t apply transformation;
toggle parameter = 1 - apply.

Command 23 - Setup anisotropy pole coordinates (parametric).

Menu line:

```
23.) Anisotropy pole: lat, lon.....................      0.00    0.00
                                                        ↑       ↑
                                                      lat     lon
```

Description: ***lat*** - latitude, (°);
lon - longitude, (°).

Dialog: …> `enter pole lat and lon 4PSI`
…> ***lat lon***

Command 25 - Perform resolution analysis (toggle).

Menu line:

```
25.) Produce resolution analysis map?..(toggle)...          0
                                                            ↑
                                                  toggle parameter
```

Description: ***toggle parameter*** = 0, no resolution analysis;
toggle parameter = 1 - perform analysis.

Command 26 - Select data format (toggle).

Menu line:

```
26.) Use new input data format R1/R2?..(toggle)...          0
                                                            ↑
                                                  toggle parameter
```

Description: ***toggle parameter*** = 0, old format;
toggle parameter = 1 - new format.
See more details in p 3.1.

Command 27 - Select ray-tracer mode (toggle).

Menu line:

```
27.) Use RAY TRACER mode?.............(toggle)...          0
                                                            ↑
                                                  toggle parameter
```

Description: ***toggle parameter*** = 0, main tomo mode;
toggle parameter = 1 - ray tracer mode.

Command 28 - Create response maps for the set of fixed points (toggle).

This command takes effect if resolution analysis is on (command **#25**), otherwise it does nothing. The set of points must be in file with the name "**SURF_POINTS"** (see section 3.5), which is resides in working directory.

*Menu line**:***

```
28.) Make response map for some points?(toggle)...          0
                                                    ↑
                                              toggle parameter
```

Description: ***toggle parameter*** = 0, no maps;
toggle parameter = 1 - create maps.

Command **29 -** Path density weights for isotropic and anisotropic parts (parametric).

*Menu line**:***

```
29.) Path density wts for iso and aniso parts..... -0.1471300     -0.1471300
                                                    ↑             ↑
                                                  wtsi          wtsa
```

Description: ***wtsi*** - coefficient for isotropic part;
wtsa - coefficient for anisotropic parts.

Dialog: …> `enter coefficients for damping: iso, ani`
…> ***wtsi wtsa***

3 Input files.

3.1 Input data - velocity measurements.

The name of input data file is defined as an argument in program command line. Data file consists of the set of lines representing single velocity measurement. There are two basic formats of the input data the old one and the new one. Old format was designed only for the first minor arc paths (orbit 1) while the new supports both minor and major arc paths (orbit 1 and 2).

3.1.1 Old format.

Each line of input data file consists of the variable number of columns. The first 7 columns should be strictly defined in the following way:
Format: free.
Column names: **`id, fi0, lam0, f1, lam1, vel_obs, weight, add_columns.`**

`id`	unique id number
`fi0`	source latitude, (°)
`lam0`	source longitude,(°)
`f1`	event latitude, (°)
`lam1`	event longitude, (°)

vel_obs	observed velocity, (km/s)
weight	weight of observed data, must be non zero
add_columns	any text beginning with space, will be stripped after input

3.1.2 New format.
Format: free.
Column names: **id, fi0, lam0, f1, lam1, vel_obs, weight, orb, add_columns.**
The new format includes additional fixed column, orb, that represents type of path orbit 1 or 2. So, the full description for each line looks in the following way:

id	unique id number
fi0	source latitude, (°)
lam0	source longitude,(°)
f1	event latitude, (°)
lam1	event longitude, (°)
vel_obs	observed velocity, (km/s)
weight	weight of observed data, must be non zero
orb	=1 - orbit 1, =2 - orbit 2, not equal 1 or 2 - orbit 1
add_columns	any text beginning with space, will be stripped after input

3.2 Contour file.

The contour defines some area on a sphere where tomorgaphic solution exists. We assume that area outside the contour is the complementing area and for this area all model parameters equal to zero and solution is simply input reference model.
The simple contour is the set of points on a sphere connected together by the parts of great circle. There are two requirements for contour construction: each point must be connected with two other noncoinciding points, and resulting curve on a sphere should not have internal intersections. When constructed, contour splits the whole sphere into two parts. The first one is internal, where solution is supposed to be computed and another one - external. So, to define simple contour we need to

setup: the point outside contour (in external part), points coordinates forming contour and links between these contour points. To setup contour we use contour file with the name defined in Menu command **#16** or use default name **`contour.ctr .`**

Consider the following example of the contour file:

```
0.7   0.7
4
0.5    0.5 #1
0.5    0.8 #2
0.8    0.8 #3
0.8    0.5 #4
4
1    2     # link 1 -> 2
2    3     # link 2 -> 3
3    4     # link 3 -> 4
4    1     # link 4 -> 1
```

The first line of the contour file contains coordinates of the point outside the contour: latitude, (°), and longitude, (°).

The next line contains the number of points forming contour (4 in our case).

Four next lines contain latitude, (°), and longitude, (°) for each points forming the contour. The third field in those lines is ignored while reading, so we can use it for point numbering (numbering from 1 by 1).

Next line is the number of links (4 too).

And, finally, links itself. Each link is represented by two point numbers have to be connected.

Actually, the contour defined by this example is whole sphere (only, if sell size greater than 1.0 degree), because outside part is a very small, and does not include any grid points.

The complex contour is the *simple contour*, say *C*, with a reduced internal part. The procedure for reducing may be described in the following way. Let us define the new set of simple contours: C1, C2, …,Cn, n >=1. Each contour Ci belongs to the main simple contour C and for each Ci, Cj , contours, $i \neq j$, intersection of the internal parts is empty, $Ci \cap Cj = \varnothing$. The internal part of complex contour will be the internal part of the main contour subtracted internal part of each contours Ci, namely, $C \setminus (Ci \cup Cj)$. We may to continue this process and create inside the external part of one simple/complex contour another new simple/complex contour in the same way. The resulting super contour is a union of both contours and the intersections of internal parts for both contours must be empty.

In case of complex situation you need to put all simple contours into the contour file, just to increase coordinate and link parts. The point defined in the first line must not belong to the internal part of the final contour.

3.3 Reference 2-D model file.

Format: free.
Column names: **`lon, lat, ref_vel`**

`lon`	longitude, (°)
`lat`	latitude, (°)
`ref_vel`	reference model velocity, (km/s)

Reference model file is on a regular grid by lat, lon with fixed integer step equal to one degree for each coordinate. Coordinate ranges: -89 <= **`lat`** <= 89, 0 <= **`lon`** <= 359. Missing points are not alluded.

3.4 1-D model for data rejection by wavelength.

1-D surface waves velocity model is used for data rejection by wavelength. It provides us with wavelength, which depends on period and type of surface wave. For example, you may take well known PREM model calculated for the fixed set of periods listed below in Table 3.4.1.

Table 3.4.1. 1-D PREM model.

```
   18
 16.0  2.85271  3.60903  3.21267  3.89173
 18.0  3.12538  3.70503  3.30155  3.98916
 20.0  3.32356  3.76737  3.40723  4.07507
 25.0  3.60025  3.85048  3.68119  4.23650
 30.0  3.72473  3.88950  3.91398  4.33775
 35.0  3.78783  3.91230  4.07599  4.39920
 40.0  3.82155  3.92776  4.17566  4.43915
 45.0  3.83938  3.94014  4.24216  4.46906
 50.0  3.84643  3.95105  4.28396  4.49255
 60.0  3.83336  3.97338  4.33114  4.52942
 70.0  3.81702  3.99680  4.35473  4.56019
 80.0  3.79525  4.02391  4.36691  4.58811
 90.0  3.77379  4.05449  4.37562  4.61547
100.0  3.75254  4.08839  4.37977  4.64217
125.0  3.70598  4.18593  4.38454  4.70871
150.0  3.66436  4.29923  4.38458  4.77658
175.0  3.62501  4.43099  4.38291  4.84670
200.0  3.59008  4.57819  4.38157  4.91906
```

Input file consists of the set of lines. The first line is the number of lines with velocity info included in file.

Format: free.
Column names: **`per, R_group, R_phase, L_group, L_phase`**

`per`	period, (s)
`R_group`	group velocity for Rayleigh waves, (km/s)
`R_phase`	phase velocity for Rayleigh waves, (km/s)
`L_Group`	group velocity for Love waves, (km/s)
`L_phase`	phase velocity for Love waves, (km/s)

3.5 File SURF_POINTS.

The file **SURF_POINTS** is not described in Menu and has fixed name. The first line of the file must include single symbol, ***`symb,`*** in the first position of line. If ***`symb`*** is **"c" xxx.crdt** file is produced, if not **"c"** doesn't. The rest lines contain coordinates ***`lat_in`***(°)**`,`** ***`lon_in,`***(°) of delta-like points, each line includes one point.
Example:

```
c
0       225.5
2       225
4.1     225
6       226
8       225
10      225
```

4 Output files.

4.1 File xxx.1

File xxx.1 defines 1x1 degree 3-D grid for the velocity in rectangular area on a sphere (lat,lon), defined by input parameters #x and #y.

4.1.1 Pure isotropic method.

Column names: **`lon, lat, velocity`**

`lon`	longitude, (°)
`lat`	latitude, (°)

velocity	estimated velocity, (km/s)

4.1.2 Anisotropy method (2ψ).
Column names: **lon, lat, vel_iso, vel_rmod, m, amp2, psi2, Acos2, Asin2.**

lon	longitude, (°)
lat	latitude, (°)
vel_iso	isotropic velocity part, (km/s)
vel_rmod	reference model velocity, (km/s)
m	velocity variations for isotropic part, **m = (vel_iso-vel_rmod)/vel_rmod**
amp2	**sqrt(Acos2*Acos2 + Asin2*Asin2)**, (km/s)
psi2	major axis direction for 2ψ, (°)
Acos2	cosine anisotropic velocity part for 2ψ, (km/s)
Asin2	cosine anisotropic velocity part for 2ψ, (km/s)

4.1.3 Anisotropy method (4ψ).
Column names: **lon, lat, vel_iso, vel_rmod, m, amp2, psi2, Acos2, Asin2, amp4, psi4, Acos4, Asin4.**

lon	longitude, (°)
lat	latitude, (°)
vel_iso	isotropic velocity part, (km/s)
vel_rmod	reference model velocity, (km/s)

m	velocity variations for isotropic part, **m = (vel_iso-vel_rmod)/vel_rmod**
amp2	**sqrt(Acos2*Acos2 + Asin2*Asin2)**, (km/s)
psi2	major axis direction for 2ψ, (°)
Acos2	cosine anisotropic velocity part for 2ψ, (km/s)
Asin2	csine anisotropic velocity part for 2ψ, (km/s)
amp4	**sqrt(Acos4*Acos4 + Asin4*Asin4)**, (km/s)
psi4	major axis direction for 4ψ, (°)
Acos4	cosine anisotropic velocity part for 4ψ, (km/s)
Asin4	sine anisotropic velocity part for 4ψ, (km/s)

4.2 File xxx.1_%_

Column names: **lon, lat, velocity_in_%**

lon	longitude, (°)
lat	latitude, (°)
velocity_in_%	(**vel_iso-vel_ref)/vel_ref**, (%),where **vel_iso** - isotropic velocity; ver_ref - reference model velocity

4.3 File xxx.res.

File xxx.res includes density for orbits R1/R2 or L1/L2.

Column names: **lon, lat, dens, dens1, dens2.**

lon	longitude, (°)
lat	latitude, (°)
dens	density for the first orbit or for the first and second together

dens1	Density for the first orbit
dens2	Density for the second orbit

NOTES The third column may be not the exact sum of the forth and fifth. The smoothing procedure is applied separately for all density columns.

4.4 File xxx.resid.

File xxx.resid includes the copy of input data (the first eight columns), residuals by time relative to inversion results and reference model, and, finally, epicentral distance.

Column names: **id, fi0, lam0, f1, lam1, vel_obs, weight, orb, res_tomo, res_mod, delta**

id	unique id number
fi0	source latitude, (°)
lam0	source longitude,(°)
f1	event latitude, (°)
lam1	event longitude, (°)
vel_obs	observed velocity, (km/s)
weight	weight of observed data
orb	=1 - orbit 1, =2 - orbit 2, not equal 1 or 2 - orbit 1
res_tomo	residual for tomo results, (s), or velocity of reference model in case of ray tracing mode
res_mod	residual for reference model, (s)
delta	epicentral distance, (°)

4.5 File xxx.azi.

File xxx.azi contains azimuthal coverage for Gaussian method only.

Column names: **lon, lat, meth1, meth2.**

lon	longitude, (°)
lat	latitude, (°)

meth1	Squared sum method, values range: 0 - 10; 10 - means the best coverage
meth2	maximum value methods. Values range: 0 - 180 degree

4.6 File xxx.rea.

File xxx.azi contains resolution analysis results.

Column names: **lat, lon, m_cone, m_gauss, amp, ncone, ngauss**.

lon	longitude, (°)
lat	latitude, (°)
m_cone	cone radius
m_gauss	Gaussian standard deviation, (σ), (km)
amp	Maximum response value
ncone	number of cells involved in cone base
ngauss	number of cells involved in Gaussian construction

4.7 File xxx.dr.

File **xxx.dr** contains covariance matrix data for isotropic matrix and for two anisortropic matrices for 2ψ. 4ψ covariance matrix is not implemented yet.

Column names: **lon, lat, c_iso, c_ani21, c_ani22**

lon	longitude, (°)
lat	latitude, (°)
c_iso	covariance for isotropic part
c_ani21	covariance for 2ψ, (cos part)
c_ani22	covariance for 2ψ, (sin part)

4.8 Files xxx.pnt.

File **xxx.pnt** contains responses surfaces for the set of points defined by ***SURF_POINTS*** file. Each surface is a response on delta-like input when resolution analysis is set up. For each surface data in file consist of the header and the corresponding xyz grid for response surface.

Structure of the header:

id, nnn, lat_in, lon_in, lat, lon, >

Where,

id	-	identification number, starting from 1 by 1;
nnn	-	number of points in grid;
lat_in, lon_in	-	initial latitude and longitude defined in **SURF_POINTS** file, (°);
lat, lon	-	coordinates of the nearest grid point to ***lat_in, lon_in,*** (°)**;**
>	-	just a symbol to simplify search of the header.

Structure of the grid point:

Lat,lon, amp

Where,

lat, lon - latitude and longitude of the grid point forming response surface, (°);

amp - amplitude at point ***lat, lon***.

4.9 Files xxx.crdt and **xxx.dr2.**

Both files are binary unformatted files. They are used in post process analysis for response surface construction.

File **xxx.crdt** contains cells coordinates and file **xxx.dr2** contains resolution matrix for all grid points.

4.10 File xxx.prot

There is an example of the protocol file. Bold lines belong to protocol, text surrounds with /* and */

only for comments.

```
/*
 * Version and method
 */
Protocol: tomo_sp_cu_s, v1.1 (Gaussian rays)
============================================================
/*
 * Observed period
 */
 PERIOD= 50(sec)
/*
 * Path to input data file
 */
 INFILE=/home/mike/src/tomo/rej_resid_data/w_rej_resid_jt_12_T_50
/*
 * Path to directory where output data will be placed and
 * output files prefix
```

```
 */
OUTFILE=XXFL12n_A2.6_2000_450_1000
model=   Y
weights=   N
dens & azimuth maps=Y
selection= N
percent map=Y
anisotropy=Y 2-psi type
rejection by %=N
rejection by dist=N
residuals= Y
covariance matrix= Y
geogr-->geoc= Y
geoc-->geogr= Y
resolution analysis = Y
input data format = Y
RAY TRACER mode = N
resolution response maps = Y
/*
 * Path to reference model file
 */
Model file name is: /home/mike/src/tomo/model/CU_Lp_50
/*
 * Path to contour file
 */
Contour file name is: contour.ctr
/*
 * Tomography parameters
 */
0PSI: alpha1, alpha2, sigma1, sigma2 2000.000  1.000  450.000  450.000
2PSI: alpha1,        sigma1, sigma2 1000.000        600.000  600.000
0PSI-prec, 2PSI-prec, 4PSI-prec    8.0    8.0    8.0
/*
 * Boundaries (window) for output maps and grid step
 */
Limits of latitude:     -89.00     89.00  Increment= 1.000 npoints=  179
Limits of longitude:      0.00    359.00  Increment= 1.000 npoints=  360
/*
 * Step of integration
 * Rayleigh/Love - type of surface wave;
 * Group/Phase velocity - type of velocity.
 */
Step of integration: 0.2500
Rayleigh/Love: R
Group/Phase velocity: P
/*
 * Path to 1-D model file
 */
Model for slowness: PREM.MODEL
/*
```

```
 * Position of pole for anisotropy, the second pole is antipode to current.
 */
 Azimus pole: lat =   0.00, lon =   0.00
/*
 * Original number of paths and paths left after possible
 * rejections by filtering procedures and contour
 */
THERE WERE  32891 PATHS; only  32891 PATHS LEFT
/*
 * Square root mean velocity, reference model suppose to be constant
 */
   SQUARE ROOT mean velocity=  4.500 km/sec
/*
 * The first timestamp. Includes human date & time (in brackets) and
 * current duration relative to previous timestamp in form:
 * hh:mm:ss  sec.
 * hh:mm:ss - current duration relative to previous timestamp,
 * sec - the same in seconds.
 */
Input     - <Sun Jun 29 11:54:58 2003>, Duration:  0: 0: 0    0
/*
 * Sizes of isotropic and anisotropic grid cells
 */
Isotropic grid cells:
 Average square of cells is:   15.5906883029 (deg**2)
 Average length of cells is:    3.9485045654 (deg)
Anisotropy grid cells:
 Average square of cells is:   40.6833937371 (deg**2)
 Average length of cells is:    6.3783535287 (deg)
/*
 * The following point must be outside the contour
 */
 Point outside contour: Lat=  0.700 Lon=   0.700
/*
 * Grids parameters for isotropic part and for anisitropic (if present)
 * for Reference Cube.
 * The next formulas are valid: nd=n_pnt1*2+1; nda=na_pnt1*2+1.
 * nd+1 is the number of grid points along cube adge (isotropy case),
 * nda+1 is the same but for anisotropic grid.
 * If contour embeds all Earth the total number of grid points,
 * nm, nma, will be: nm=6*nd*nd+2 (isotropic grid), nma=6*nda*nda+2 (aniz).
 * Also, aniz =0 - for 0psi, =1 - for 2psi, =2 - for 4psi.
 */
 n_pnt1 =  10 na_pnt =  6
 Cells:  2648, nm= 2648, nd+1= 22, aniz= 0
 Cells:  1016, nma= 1016, nda+1= 14, aniz= 1
/*
 * Normalization weights relative 2x2 grid cell size
 */
Normalization factors: Isotr -  1.95238, Aniz:  3.15385
```

```
/*
 * Total number of path, for the first orbit and for the second
 */
   Number of Paths=  32891
 R1 traces: 23435 R2 Traces: 9456
/*
 * The number of Gaussian integrals involved in tomography
 */
 Number of integrals are:  32891
/*
 * Square root of initial residuals by time and velocity.
 * Weighted residuals are printed in slashes, for example, / 17.370/.
 */
SQRT of initial residuals dispersion are:   17.370 sec /   17.370/
SQ_RT of initial velocities dispersion are:   0.02487 km/sec
/*
 * Timestamp. Duration is the integrals computation time.
 */
Integrals  - <Sun Jun 29 12:06:29 2003>, Duration:  0:11:31   691
/*
 * Azimuthal coverage parameter, maxima method
 */
  MAXIMUM of asimutal parameter is :   140.959457
/*
 * Timestamp. Duration is the Gaussian smoothing computation time.
 */
Gauss      - <Sun Jun 29 12:06:32 2003>, Duration:  0: 0: 3    3
/*
 * The main matrix is constructed. Total number of matrix elements
 * and number of elements equals to zero.
 */
 Matrix is done
 Matrix elements: 21996100 Zero are: 6960
 Total number of matrix lines are 37571
/*
 * Maximal values for the total density and for the first and second
 * orbits, separately
 */
 Maximum intersections/cell are:   944.618286 R1:  762.193542 R2:  526.741516
/*
 * Matrix trace divided by the size of diagonal
 */
 Isotropy   Trace/Size F  13148305.0000
 Anisotropy Trace/Size F  12554453.0000
/*
 * Timestamp. Duration is the matrix inversion time.
 */
Inversion  - <Sun Jun 29 12:10:03 2003>, Duration:  0: 3:31   211
Matrix condition =    0.170139E-03
/*
```

```
 * Timestamp. Duration of different operations over huge matrices,
 * when resolution analysis, covariance matrix computation are set.
 */
Inv. matrx - <Sun Jun 29 12:14:35 2003>, Duration:  0: 4:32   272
G-1 * G    - <Sun Jun 29 12:22:44 2003>, Duration:  0: 8: 9   489
Matrix OK. - <Sun Jun 29 12:22:48 2003>, Duration:  0: 0: 4     4
/*
 * Timestamp. Duration is the resolution matrix computation time.
 */
Res_matrix - <Sun Jun 29 12:26:37 2003>, Duration:  0: 3:49   229
/*
 * Next line contents the number of elements in resolution matrix, r,
 * and covariation, c, between transpose elements:
 * c=sqrt(sum((r(i,j)-r(j,i))**2)/(n-1)), where i != j, n - number of
 * non diagonal elements.
 */
 Matrix elements= 7011904, Maximal deviation =  0.00117079378
/*
 * Timestamp. Duration is the resolution analysis time.
 */
Res_analys - <Sun Jun 29 12:26:42 2003>, Duration:  0: 0: 5     5
/*
 * Square root of final residuals by time and velocity.
 * Weighted residuals are printed in slashes, for example, / 14.266/.
 */
SQRT of final residuals dispersion are:   14.266 sec /   14.266/
SQ_RT of final velocities dispersion are:   0.02149 km/sec
/*
 * Timestamp. Duration is the final residuals computation time.
 */
Velocities - <Sun Jun 29 12:32:10 2003>, Duration:  0: 5:28   328
/*
 * Average velocity (isotropic part), correction for sphere is applied.
 */
 average velocity =  4.50669 (km/sec), npoints=  64440
```

5. Technical notes.

5.1 File tomo.h

While compiled, **tomo_sp_cu_s** program uses include file **tomo.h,** which contains four parameters and common block variables. One parameter, **NINP**, defines the maximal number of measurements in input data file. Program can read the amount of data less or equal than **NINP**. The rest of data is rejected. The next three parameters, **NRAZ, NRAZA, NAZIPL**, define the size and amount of memory statically allocated for the huge array **f** (***swt_sph.f*** module), which is used for inversion. It is not possible to increase the size of array after compilation. So, before running program be sure that you do not reject data and have enough memory allocated (see section 5.2 for more detail). For example, the beginning of the **tomo.h** file looks like:

```
c****************************************************************
c  NINP  -  max number of input rays
c  NRAZ  -  max number of isotropic grids
c  NRAZA -  max number of anisotropic grids
c  NAZIPL - 1, 3 or 5. 1 - 0PSI, 3 - 2PSI, 5 - 4PSI
c****************************************************************
     integer*4  NRAZ, NRAZA, NINP, NALPH, NAZIPL, IAZIM
     parameter (NRAZ=10100,NRAZA=1020,NINP=100000,NALPH=4,NAZIPL=3,IAZIM=10)
c----------------------COMMON /charac/-----------------------------
     character*160 namout,namein,outfile(NALPH)
     character*18 com
     common/charac/namout,namein,outfile,com
            . . . . .
            . . . . .
            . . . . .
```

To compute the actual number of members **N** and the amount of allocated memory **M** in bytes for array **f** use the next formulas:

N = (NRAZ+NRAZA* (NAZIPL-1))*(NRAZ+NRAZA*(NAZIPL-1)),

M = N * 4

For our example **N = 147379600** and **M = 573518400** bytes.

5.2. System/program requirements.

Memory management. Usually the maximum alluded program data size is defined by OS and compiler. For the most OS such as Linux, Solaris, SunOS this size is close to 0.9 - 1.0 GB. To increase this size it is necessary to upgrade OS parameters and recompile the kernel. If it does not help, try to use contour technique - split the area of investigation on a few parts and make tomography for each part separately.

Disk space management. If you toggle resolution analysis or covariance matrix computation you need to provide enough disk space in working directory. It should be enough to store double size of array **f** in case resolution analysis, and triple size of array **f** in case covariance matrix or both.

5.3 Error messages.

1.(R0010) File ***file_name*** does not exists

Program tries to open non existing file or file with wrong name. Program terminated. Add required file or correct file name and start program again.

2.(R0011) Error in model ***fi*=±dd.ddd, *lam*=±ddd.ddd**

Model file doesn't defined at point with coordinates ***fi ,lam***. Program terminated. Correct reference model file.

3.(R0012) SIZE OF **NRAZ** IS TOO SMALL ***nnn*** SHOULD BE: ***mmm***

Not enough memory allocated for matrix **f** (isotropic part). Program terminated. Decrease size of isotropic cell (command **#10**, field ***si***) or increase the value of parameter **NRAZ** to ***mmm*** in **tomo.h** file and recompile program.

`4.(R0013) SIZE OF NRAZA IS TOO SMALL` ***`nnn`*** `SHOULD BE:` ***`mmm`***
Not enough memory allocated for matrix **f** (anisotropic part). Program terminated. Decrease size of isotropic cell (command **#10**, field ***sa***) or increase the value of parameter **NRAZA** to ***mmm*** in **tomo.h** file and recompile program.

`5.(R0015) 1-D model period is out of range`
Expand 1-D model or change period.

`6.(R0098) END of RAY TRACING`
Successful end of **tomo_sp_cu_s** program (ray-tracing mode).

`7.(R0099) COMPUTATIONS FINISHED`
Successful end of **tomo_sp_cu_s** program.

`8.ATTN l=` ***`nnn lat lon`*** `imax=` ***`mmm lat1 lon1 amp`***
Expected position ***`lat lon`*** of maximum response function is not valid. Actual maximum position is at point with coordinate ***`lat1, lon1. amp`*** - amplitude at ***`lat1, lon1. nnn`*** and ***`mmm`*** are number of columns in matrix **f** for expected and real position. Program continue to run**.**

6. References.

Barmin, M.P., M.H. Ritzwoller, and A.L. Levshin, A fast and reliable method for surface wave tomography, PAGEOPH, **158**, n.8, 1351-1375, 2001

Ritzwoller, M.H., N.M. Shapiro, M.P. Barmin, and A.L. Levshin, 2002, Global surface wave diffraction tomography, J. Geoph. Res., **107**, B12,2335,doi:10.1029/2002JB001777,2002