GEOFORCE:
A programme package for the study of viscoelastic deformations of the Earth

Hans-Peter Plag
8720 Rainbow Trout Ct.
Reno, NV 89523, USA
Phone +1-775-747-6185
E-mail: hpplag@unr.edu

Version 1.1
June 3, 2009

.

Contents

Chapter 1

Introduction to the program package GEOFORCE

GEOFORCE is a programme package for studies of the visco-elasto-dynamical properties and the exogenic deformations of the Earth (or, in general, of any celestial body). The package was developed and is currently maintained by the "Global Geodynamics Group" at the Institute of Geophysics of the University of Kiel.

GEOFORCE is based on three main utilities, namely DIALOG (see Chapt. 2), a programming utility for the design of interfaces between men and programmes, ACCESS (see Chapt. 3), a programming utility for access to data files and data handling, and PLOTSOFT (see Chapt. 4), which is the kernal for the graphical output of GEOFORCE. These basic concepts were developed by Plag (1982, 1988), and these are discussed in Part I of this documentation. Furthermore, GEOFORCE assimilates two programme packages also largely developed by Plag (1982), namely, POTS and PRESENT (see below).

Based on these utilities, GEOFORCE is an easy to handle and effective research tool. From a research point of view, GEOFORCE separates into four main parts.

RHEOLOGY is the part used to edit and study the Earth model or, in principal, any other model of a celestial body (see Part II). The editing of the model includes the elastical, viscoelastical, and thermodynamical parameters as well as the geometric structure. The system's response is studied by calculating free oscillations, or Love Numbers for both, body or surface forces. This part also includes the calculation of Green's functions for loading.

For given exogenic forcing, EXTERN is used to calculate observable quantities, such as gravity changes, or those quantities derivable from observations, such as expected Geoid changes (see Part III). The results of these calculations may then be compared with observations, and, clearly, this comparision is the only test for the quality of the models and the theory used in the calculations.

The model quality will, of course, depend on the database used in the test. POTS (Processing Of Time Series) is designed for the handling and the analyses of observational or synthetic data (see Part IV).

In most cases, a graphical representation of the results is an important tool in the assessment of the results, both from the theoretical calculations and from the processing of the observations. Therefore, PRESENT is integrated into GEOFORCE to supply some powerful graphic tools (see Part V). PRESENT is based upon the PLOTSOFT utility, mentioned above.

The documentation of GEOFORCE presently is not finished, i.e. for a number of topics, no documentation is available. Nevertheless, those parts included in this document give a rather complete picture of the overall design of GEOFORCE, how it works, and how it is to be used.

Part I

The basic utilities of GEOFORCE

Chapter 2

DIALOG: The interface between man and programme

DIALOG is a programming utility for the design of the interface between men and programmes. DIALOG was especially designed

      - to be machine-independent,

      - to prevent input errors as far as possible,

      - to supply routines for all standard I/O tasks,

      - to be as stable as possible in its performance,

      - to make input as comfortable as possible.

The design of such an interface was stimulated by the sometimes tedious experience each programming scientist will eventually make, namely that getting a new computer, exchanging programmes with collegues, or even moving to a new institute always cause a lot of work in adopting programmes to the new environment. During more than two decades of working in scientific programming quite a number of routines were designed for all kinds of standard tasks encountered nearly every day in working out new programmes. These routines eventually formed the basis for creating DIALOG.

DIALOG uses the concept of menus, with each menu providing a task-oriented environment with a well defined set of commands. Usually, the number of commands within a menu is of the order of ten. In addition to these specific commands, a set of task-independent commands, which are frequently needed on any level of the programmes, is always available. This so-called COMMON menu is described in detail in Sect. 2.1.6.

The core of DIALOG is the Command Line Interpreter (CLI). The CLI is discussed in Sect. 2.1 below. The performance of the CLI may be manipulated by several of the commands supplied by the COMMON menu.

A number of certain menus are available in most of the programmes using DIALOG. These menus supply service functions, allow for access to the results of any calculation, help to manipulate time and date parameters a.s.o. All the menus that are an intrinsic part of DIALOG are discused in the present chapter, with a separate section devoted to each menu. In task-oriented programming, some commands ought to be present in most menus, having, however, a slightly environment-dependent operation. Their general use and structure are also discussed here.

2.1 The command language interpreter (CLI)

2.1.1 CLI command syntax

The CLI reads and interprets Command Language (CL) strings. The CLI interprets strings following an exclamation mark as comment. Otherwise, the syntax of a CL string has the format

command_string := command [sub_string|parameter_string]

Here, command is a unique first letters abbreviation of a full command name from the current menu or the COMMON menu. Commands may be either names of submenus or they may denote specific tasks of the current menu that are carried out immediately. The number of characters required for uniqueness of the abbreviation depends on the set of commands available in the current menu including the COMMON menu. The vertical bolt stands for an exclusive OR, i.e. the command is followed by a blank and either a parameter_string or a sub_string. The square brackets around an item denote its optionality. If the command is a name of a submenu, then a sub_string may be given, which is passed on to the lower menu (see below). If the command is not a name of a submenu a parameter_string may be given.

The exact structure of the parameter_string depends on the specific command, and a description is given for each command separately. However, the syntax of the parameter_string has the form

parameter_string := [parameter]{,[parameter]},

where the braces denote the optional repeatability of the enclosed element.

Parameters may be strings, numerical parameters, dates, logical switches, or equations.

The syntax of strings depends on the number of parameters, and in some cases, a string may not contain a comma, or, within a sub_string (see below and Sect. 2.1.3), no blanks. In the latter case, blanks may be denoted by the underline character.

The syntax of a numerical parameter depends on its data type, which may be integer, real, double precision or complex. The CLI interprets numerical parameters as algebraic expressions. These expressions may include variable names (see Sect. 2.1.4). The evaluation of expressions are performed in double precision and converted into the required data type. Conversion to integer is done to the nearest integer. The algebraic expressions may use the mathematical operators $+,-,*,/,\hat{}$, with the last operator denoting potentiation.

Dates have the syntax yyyy-mm-dd-hh-mm-ss, where any of the parts may be omitted, as long as the meaning is clear. Valid dates are 1985- - -12-15, 1900-1-01-, and -12-31-. The missing parts are substituted by default values, which depend on the context. In most cases, dates are defined by the data read in, and missing parts are left unchanged. All parts are checked for consistency, i.e. -2-30-60 would be an invalid date, since February doesn't have 30 days and a day doesn't have 60 hours.

Logical switches are generally set by either 'YES' or 'NO' or any abbreviation of these. In this documentation, a logical parameter is simply referred to as 'switch'.

Equations may occur in two different forms. In the EQUAL command (see Sect. 2.1.6), equations have the syntax

equation := internal_variable=string

while for matrix operations the syntax is

equation := column_name=algebraic_expression.

Sub_strings are command_strings to be passed to a submenu (see Sect. 2.1.3). Their syntax is

> sub_string := short_command{ short_command},

where

> short_command := command[=parameter_string|[(]sub_string[)]].

In this case, command is not an abbreviation of a command name of the current menu but rather of a command name in the submenu called by the command_string. This so-called shortcut call is explained in more detail in Sect. 2.1.3. The brackets around the sub_string in a short_command are required if the sub_string contains any blanks.

2.1.2 Interactive and Batch modes of the CLI

The CLI discriminates between two basic modes, namely the Interactive Mode (IM) and the Batch Mode. In IM, the CLI assumes a person to physically sit in front of a screen, thus being available to answer questions in all cases where incomplete information was entered. In BM, no such person is assumed to be present, and, in case of incomplete information, the programmes have to decide what to do. Normally, default values are defined to be used in such cases. However, sometimes, warnings or even error messages will be issued whenever a situation results where missing information can not be securely substituted in BM.

It should be noted, that IM and BM do not mean, that the programme is necessarily running interactive or in an submitted job, respectively. Both modes can be usefull during interactive sessions, but within a submitted or queued job, BM should preferably be used. Using BM in an interactive process may cause problems to an unexperienced user, because it supress the prompt.

2.1.3 Shortcut calls to submenus

In many calls to a submenu, only a certain sequence of a few commands will be entered in this menu, and a return to the calling menu after that is already certain from the start. In these cases, a shortcut call to the submenu is usefull.

The following examples illustrate the use of the shortcut call. A call to EDITIA (see Sect. 2.3.2) could have the following form

```
> editia
> number 4
> all 40,30,20,10
> show
> end.
```

Using the shortcut version, this would look like

```
> editia num=4 all=40,30,20,10 show
```

Both examples have identical effect. In IM, if an error in one of the commands is detected during a shortcut call, CLI asks for a renewed complete input of the erroneous command, including the parameters. That is, typing 'rall=40,30,20,10' instead of what would be correct, would result in a request for typing this anew and correctly. In BM, this error would result in a number of subsequent errors, and most likely, the job would have to be done again.

2.1.4 Using variables in commands

The CL provides a set of variables, which can be used to transfer string values from one menu or procedure file level to another level, and they are helpful in writing procedures with parameters that are easy to change.

The names of these variables range from $A to $Z. The EQUAL command (see Sect. 2.1.6) is used to equate a variable to a string. Whenever a $ appears in a parameter_string, CLI interprets this together with the next letter as a variable name, and the variable name is replaced by its string value before any further processing of this parameter_string is done. Variables may only be used after being set to a value with an EQUAL command. Undefined variables are marked with '−>undefined'.

In addition to the variables $A to $Z, the variable $$ is defined to allow input of a $-sign, i.e. the variable $$ always has the value $.

Only the parameter_string or sub_string of a command line is checked for the appearence of variable names. However, if a command name is included in a sub_string (short cut call, see below), it may contain variable names. In this way, procedures with alternative actions may be created.

Algebraic expressions may contain variable names. However, the user is responsible for assigning correct strings to the respective variables. The string assigned to a variable may be an algebraic expression. It should be noted, however, that variables are used in string-replacement mode, only, which excludes any direct or indirect recursive expression like

```
> EQUAL $S=$S+1
```

or

```
> EQUAL $A=$B+1
> EQUAL $B=$A+$C.
```

2.1.5 Evaluation of numerical parameters

Any command requiring numerical parameters in the parameter_string accepts arithmetic expressions. An expression is valid if it follows the usual rules of writing arithmetic expressions in programes. However, using variables in expressions, it should be noted, that only text replacements are being made. Thus, the following example

```
> EQUAL $A=10.+3
> EQUAL $B=-19.
> ...
> number $A/$B
```

results in the invalid expression

```
> $A/$B = 10.+3/-19.
```

because the combination /- is not allowed. A correct way to write the example would be:

```
> EQUAL $A=(10.+3)
> EQUAL $B=(-19.)
> ...
```

Table 2.1: COMMON (V1.3): The Common menu

Command	Description
EQUAL	Set one of the variables from $A to $Z
VAR_SHOW	Show list of all variables
VAR_SAVE	Save variables to a file
VAR_LOAD	Load variables from a file
DO_FILE	Read directives from file and execute them
LOOP	Do a loop over a given time interval
BATCH	Switch between interactive and batch mode [N/Y]
END	Return to next higher menu
HELP	Display current menu (include common menu [N/Y])
SHOW	Show status information of the current menu
INFO	Display additional information related to current menu
P_DOC	Display documentation concerning current menu
C_DOC	Display contents of documentation
DEBUG	Set debug level (0: no deb., <10: increas. deb.)
MENU	Set menu output level (0 - 5)
PROTOCOL	Switch protocol (on list file) on or off [Y/N]
TIME_USED	Show cpu time used up to now
COMMENT	Enter a comment to be written to list file

```
> number $A/$B
```
would result in the valid expression
```
> $A/$B = (10.+3)/(-19)
```
which in addition to correcting the syntactical error also takes into account the consequences of the text-replacement mode.

Expression are always evaluated in double precision mode and converted down to the required precision of the specific parameter. Real to integer convertion is done to the nearest integer. Thus, if an integer number is required, the above example would result in a nearest integer of -1.

2.1.6 The COMMON menu

The COMMON menu is available at any point of the program. The structure of this menu is given in Table 2.1.

EQUAL is used to assign a value to one of the internal variables. The use of these internal variables is discussed in Sect. 2.1.4. Examples of the EQUAL command are
```
> EQUAL $F=input.file
```
and
```
> EQUAL $O=1900-01-01-0-0.
```
With VAR_SHOW, the list of the variables from $A to $Z is displayed. The commands VAR_SAVE and VAR_LOAD can be used to transfer variables from one programme to another, thus providing

15

a means for designing procedures, with a variable definition at the beginning of a sequence of programmes.

The DO_FILE command has one parameter, which is the name of a procedure file. A DO_FILE command assigns the file given as parameter to the input channel for commands, and the programme continues to read commands from the file until the End Of File is reached. There is no default extension for the procedure file, i.e. the full name must be given. The procedure file must contain a valid sequence of commands of the current programme, including commands of the COMMON menu. Procedure files may contain further DO_FILE commands, but recursive calls result into an error condition. Examples of the command are

```
> do diagram.pro
```

and

```
> EQUAL $P=procedure.pro
> ...
> do $P
```

The LOOP command is specifically designed to help in moving analyses. An example is given by

```
> loop 1900,1990,10,2,years,loop.pro
```

which will perform a moving analysis with a window length and a shift of 10 years and 2 years, respectively, in the interval from 1900 to 1990. For each window, the procedure given in loop.pro will be executed until the command INCREMENT is encountered. INCREMENT has a switch, which decides what happens after the end of the time interval is reached. If the switch is set to YES, the execution will continue with any commands in loop.pro given below the INCREMENT statement.

The switch BATCH is used to activate or deactivate the BM. The latter case automatically activate the IM of the programmes (see Sect. 2.1.2).

The command END results in leaving the current menu and returning to the next higher menu level. No specific command is supplied to terminate a programme, and if END is entered in the main menu of a programme, the programme terminates its execution.

SHOW displays information about the current status within the present environment. The output of SHOW depends on the menu. Normally, in main menus SHOW includes the file status, the time parameters, any select status, and the status of the data matrix. In many cases, the input parameters set within a DEFINE menu are also included. The additional information strongly depends on the programme, and its size is highly variable. In submenus, SHOW displays information of particular interest within the environment of this menu. In many case, the information displayed by a submenu is more detailed than what is displayed in the higher level menu, and sometimes only available within the submenu.

In a number of menus, additional information is available about some or all of the commands of the menu. Especially in cases where certain tasks may be performed by a special sequence of commands, this is included in this information. INFO is used to display the available information.

With HELP, the current menu can be displayed on screen (in IM) or included in the log-file (in BM). In IM, the COMMON menu may also be shown.

Depending on the installation, documentation of programmes running under DIALOG may be available on-line. If so, calls to the on-line documentation utility are made by the P_DOC command of the COMMON menu. The command P_DOC may be issued with or without a parameter. Calls to

Table 2.2: Parameters of the P_DOC command

PARAMETER	Item displayed
?	Description of the documentation utility
*	Same as ?
DIALOG	General description of the DIALOG utility
CLI	Description of the Command Line Interpreter
SYNTAX	The syntax of CLI command lines
BATCH	Use of Batch and Interactive Modes
SHORTCUT	Description of shortcut commands
VARIABLES	How to use internal variables of the CLI
COMMON	Documentation of the COMMON menu

this utility are treaded in a manner depending on the parameter given. Entering the command P_DOC with no parameter results in displaying the documentation of the active menu. To get additional information about a specific command of the current menu, this command has to be given as parameter of the P_DOC command. If there is no information for the current path plus the parameter, then the doc-file is searched for an entry with either '*.'+parameter or '*.'+current menu.

A list of all documentation items is displayed using the command C_DOC. In this list the items are numbered. To access any documentation item directly by P_DOC, the paths to this item ought to be entered as parameter augmented by a leading '>'. An more comfortable alternative is to use the item's list number. Examples are:

```
> p_doc >how_to_use_this_doc
> p_doc #1.
```

Both commands have the similar result.

Table 2.2 compiles specific parameters of the P_DOC command together with the items being displayed in response to them.

An example of the P_DOC command is

```
> P_DOC CLI.
```

All programmes running under DIALOG provide a debug utility. The amount of debug messages issued during run-time depends on the debug level, which can be changed by the DEBUG command. Valid debug levels are between 0 and 10, with debug increasing with the level. Thus, if the level is 0, no debug messages are issued. The debug utility is of special value during programme development and testing, but has also its usefulness in understanding obviously weird results. In most cases such results are due to misunderstanding the programme and not to actual errors, and these cases are usually clarified by a debug level of 1 or 2. Debug levels larger than 3 or 4 may produce large amounts of output including the contents of arrays a.s.o.

Prior to prompting for a new command line, DIALOG may issue part or all of the current menu. The amount of information is selected by the MENU command, with the effects summarized in the Table 2.3. In most cases, the menu level 1 will be sufficient.

DIALOG provides a protocol utility, which is activated by the switch PROTOCOL. If the protocol is activated, all command lines including their interpretations and resulting outputs are written to the list file of the programme.

Table 2.3: Menu levels and their effect

MENU LEVEL	Action
0	No menu output prior to prompting
1	Output of commands prior to first prompting within the menu
2	Output of full menu prior to first prompting within the menu
3	Output of commands prior to prompting
4	Output of full menu prior to prompting
5	Output of full menu including comments on commands

TIME_USED displays the cpu-time already consumed by the current run of the programme.

With COMMENT, additional comments may be written directly to the list file. The list file of a programme always contains information on the data used or the actions performed. However, in order to avoid later misinterpretations it is often desirable to further comment the results included in the list file. After issuing the COMMENT command, all subsequent lines are interpreted as comments, until a line solely with 'END' (with no leading blanks) is encountered.

2.2   Menus frequently available or used for standard tasks

DIALOG offers a set of menus, or menu structures, for standard tasks required in many programmes. One module, namely the FILES module, controls all the opening of input and output files and supplies a number of routines that allow different parts of the programs to get information about the status of the files and attributes as well as to access the files. The menu FILES (Section 2.3) is the interface to this module.

Another module controls any input of date and time and supplies routines for the manipulation of these quantities. The menu SET_TIME is the interface between the CLI and this module.

Editing of one-dimensional or two-dimensional data arrays for different data types is another standard task, and comfortable menus are supplied by the generic menus EDITXX and EDIXAN, respectively.

2.3   FILES: Open input or output files

All file opening activities are concentrated in the FILES menu. The actual content of the menu depends on the programme, but Table 2.4 gives a typical example of a file menu of a programme requiring two input files and one output file. These files are denoted by their logical file names, that is the items fo the menue.

Within FILES, SHOW displays the current file status. INFO can be used to get some additional information on how to use FILES. The command CLOSE allows to close files by their logical names. Thus, in the example in Table 2.4, the results file can be closed by

```
> files close=results.
```

Table 2.4: FILES: Open data files for input and output

Command	Description
INPUT	Open input data file
MODEL	Open input file for model
RESULTS	Open output file for the results

Table 2.5: SET_TIME: Define time parameters for following operations

Command	Description
ORIGIN	Set time origin
UNITS	Define time units (SECONDS, MINUTES, HOURS, ...)
INTERVAL	Define time interval to be used
DELTA	Set step width to shift interval
SHIFT	Shift interval by delta
ZERO	Set time origin to begin of interval
RESET	Reset time interval and origin

2.3.1 SET_TIME: Set time parameters

The definition of time parameters valid for the subsequent operations is done by SET_TIME (see Table 2.5). In this menu dates must be entered according to the syntax: YYYY-MM-DD-HH-MM. A date may also be abbreviated, for example by giving only year and month. In such cases, the values of the omitted items remain unchanged.

ORIGIN sets the time origin, which defines the time reference for the subsequent operations. By default, the time origin is usually that given in an input data file, which can be overwritten using the ORIGIN command. Reading in a time series, time is transformed into differences between time origin and sampling time.

The time units used are usually those supplied in an input data file. UNITS may be used to set the actual time units to any other desired unit, which may be one of SECONDS, MINUTES, HOURS, DAYS, WEEKS, MONTHS, and YEARS. The lengths of a month is determined as 1/12 years, with a year having 365.249 days.

With INTERVAL, the time interval for data selection is set. This command requires two dates as parameters, which define the begin and the end of the time interval to be considered during any subsequent operation.

For moving analysis like moving averages, the time interval has to be shifted repeatedly by a fixed amount. For this purpose, the menu includes the commands DELTA, SHIFT and ZERO. The step width of the shift is set by DELTA, where delta requires a numerical parameter denoting the step width in the current time units. SHIFT actually shifts the current time interval by the amount specified by DELTA. With ZERO, the time origin is set to the begin of the current time interval.

Sometimes it is necessary to delete the time interval, time origin or time units set by the user. The RESET command sets all these quantities into the NO-SET status, enabling the programme to use the defaults again.

Table 2.6: EDITXX: Editing of a one-dimensional array

Command	Description
NUMBER	Set number of active array elements
ALL	Enter all active elements
CHANGE	Change one element [index,value]
DELETE	Delete one element [index,decrease]
INSERT	Insert a new element [index,value,increase]

2.3.2 EDITXX: Editing one-dimensional data arrays

DIALOG supplies a set of menus used to edit data arrays. The title EDITXX comprehends four different editing menus, namely EDITIA, for editing arrays of the data type Integer, EDITRA, for editing arrays of the data type Real, EDITDA, to edit arrays of the type Double Precision, and EDITYN to edit arrays of switches. There is also a similar menu EDITTX to edit character strings, however, this menu does supply some additional commands for text editing (see Section 2.3.3).

The specific name of EDITXX is determined in the calling menu and passed to EDITXX. For all data types, the menu has the structure given in Table 2.6.

The INFO command displays an explanation of the physical meaning of the values to be edited, while the SHOW command displays the current content of the array.

With NUMBER, the number of active elements in the one-dimensional array is specified. ALL is used to enter all active elements, and for each element, the current value is the default.

CHANGE is used to enter the value of one specific element. The parameter string of CHANGE contains two parameters, the first one being the index of the element, and the second one being the new value for the element.

DELETE and INSERT are used to delete existing or insert additional elements, respectively. The parameter string of the DELETE command has two parameters, with the first one being the index of the element to be deleted, and the second one being a switch, which determines whether or not the number of active elements is reduced by one. In any case, the contents of the elements behind the deleted one are shifted foreward by one position, and the last one remains empty. The INSERT command is similar to DELETE. It has three parameters, namely, the index of the element, where the new element has to be inserted, the value of the new element, and a switch, which determines whether the number of active elements is increased by one. Here, too, all elements including the one with the index given are shifted backwards by one position, and the last element may or may not be lost, depending on the switch value given.

An example of an editing session for integer data could be

```
> number 4
> all 3,7,9,-1
> show
> ...
> change 4,1
> insert 1,5,y
```

20

Table 2.7: EDITTX: Editing text (character) arrays

Command	Description
NUMBER	Set actual number of names
ALL	Enter all names
CHANGE	Change a name [index,name]
DELETE	Delete a name [index,decrease Y/N]
INSERT	Insert a name [index,name,increase Y/N]
SPLIT	Split a name into two
REFORM	Reformate the text
BLOCK	Perform block formatting of text array

> end

In some cases, the number of active elements is determined by the tasks of the calling menu, and in these cases, the number of active elements may not be changed within the editing menu. Then, NUMBER as well as the respective options of DELETE and INSERT are disabled.

2.3.3 EDITTX: Editing text (character) arrays

The EDITTX menu has three commands in addition to those described above for the other EDITXX menus. namely the SPLIT, REFORMAT and BLOCK commands. With SPLIT, a string can be splitted into two, thus allowing to insert text in a line without repeating the other text. The REFORMAT command serves to format the text into a block, where each string separated from the rest by a leading and trailing blank is considered as a word. REFORMAT fills each line, starting from the first one, with as many words as possible. BLOCK reformats the text into block with the text being adjusted to the right and left side margin. For that, BLOCK fills each line at all space between words with blanks so that the last non-blank character is aligned with the right side margin

2.3.4 EDIXAN: Editing two-dimensional data arrays

The EDIXAN menus are used to edit two-dimensional data arrays. The rows of the array are considered as tuples, while the columns contain the different components of each tuple. The EDIXAN menus have the same structure as the EDITXX menus, where each item is now a tuple instead of a simple array element. As is the case for the EDITXX menus, the specific name of a menu is passed from the calling menu. Three different versions of EDIXAN exist, namely EDIIAN, for editing arrays of the data type Integer, EDIRAN, for editing arrays of the data type Real, and EDIDAN, to edit arrays of the type Double Precision. For all these data arrays, the menu has the structure given in Table 2.8.

The use of the commands is the same as decribed for the EDITXA menus (see Sect. 2.3.2). However, in addition to NUMBER for setting the active number of tuples, the command DIMENSION is used to define the number of active columns of the matrix array, i.e. the number of components, each tuple has. ALL requires the elements to be entered as tuples, starting with the first one. The commands CHANGE, DELETE, and INSERT all apply to whole tuples instead of single elements. To enter the

Table 2.8: EDIXAN: Editing of tuples

Command	Description
NUMBER	Set number of active tuples
DIMENSION	Set number of active columns
ALL	Enter all tuples
CHANGE	Change one tuple [index,values]
DELETE	Delete one tuple [index,decrease]
INSERT	Insert a new tuple [index,values,increase]

Table 2.9: SELECT (Vn.n): Select items from a list

Command	Parameter
Task/Option 1	Perform task/Select option 1
...	...
Task/Option n	Perform task/Select option n

two geographic position (110E,24N) and (120E,25N) into an array would be achieved by e.g.

```
> dim 2
> num 2
> all 110,24,121,25
> chan 2,120,25.
```

2.3.5   SELECT: Select items from a list

In most programmes, a list of different, though not necessarily alternative, tasks is available. In addition, several alternative options have to be specified. The SELECT menu is used to choose those tasks to be performed within a session or part of it, and to specify the options. Of course, the specific name of the menu as well as the list of tasks and options depends on the calling menu.

In each call to the SELECT menu, an explanation of the different tasks and options and any peculiarities of them is supplied. The INFO command results in displaying this information. Likewise, the SHOW command displays the current status of the switches for each of the tasks or options.

2.3.6   SPECIFY: Specify one item from a list

Whenever exactly one item from a list of possible items has to be chosen, SPECIFY is used to accomplish that (see Table 2.10).

SPECIFY is rather similar to SELECT, however, only one item may be chosen. Both, INFO and SHOW are available here, too. Again, INFO displays an explanation of the different items as supplied by the calling menu, while SHOW displays the currently selected item.

Table 2.10: SPECIFY (Vn.n): Select exactly one item from a list

Command	Description
Item 1	Select item 1
...	...
Item n	Select item n

Chapter 3

ACCESS: an easy access to data files

3.1   The basic concept of ACCESS

The ACCESS utility provides access either to external data files or the data base files. The data of different files may be mapped into an internal data matrix, which may be modified subsequently. Each column of the data matrix represents a parameter while each row carries one sample for all columns. Normally, the first column is assumed to contain the coordinates of the sample points (i.e. the sampling time, a sampling coordinate, or the number of the sample).

ACCESS can handle several different types of data file structure. The available structures are described in the next table.

The standard data files have a ten record header containing a standardized description of the data. In these files, the file type is denoted by a key number in the header. These files may be written as formatted (i.e. plain ASCII) or unformatted (i.e. binary). A standard data file may contain either a single, equidistant series of a parameter (usually a time series), or a set of (X, Y1, ..., Yn)-tuples. The actual format of the data is given in the header and is largely free. The history included in some files accounts for any modifications applied to the data.

Seqmented data files have, for each seqment, a descriptive part and a data part. These files are usefull to combine related information of different structure or format in one file. The segmented files are also used to store data to be included in a larger data base. The data base inventory file has a special format created by the programme ACIDE, and these files organize large numbers of different data entities.

Table 3.1:

Value	File type
0	Undefined and task dependent type, no header assumed
1	Standard data file with a 10 line header
2	Standard file including a history
3	Advanced segmented data file
4	Data base inventory file

Table 3.2:

EXTRACT (V.1.3)	The EXTRACT menu
DIMENSION	Set dimension of data vector and open work file
MAP	Define mapping of current input into data vectors
LINKS	Define links between different files
COMPONENTS	Define relation between current input and data vectors
ERRORS	Define relation between input and error vector
CONSTRAINTS	Define constraints for vector components
STEP	Set step for input data to be used
OFFSET	Set number of first value to be used
READ	Read selected data into data vector
SORT	Sort data according to increasing x

To create a data matrix within a programme, the EXTRACT menu is used, and this menu also maps data from external input file into the current data matrix.

A large set of different modifications of the columns or rows of the data matrix may be performed within TRANSF. Besides editing the data, performing standard calculus or vector analysis operations or statistical tests, the data matrix may also be filled with newly created data.

Also included in the ACCESS utility are a number of subroutines for the actual access to the external files or the data matrix, which are described in more detail in the tecnical programme documentation.

3.2   EXTRACT: Mapping data into the data matrix

The EXTRACT menu is used to read data from (several) input file(s) to a data matrix, which is the base for the subsequent computations. The matrix consists of n columns and m lines. Each column represents one data type while each row is interpreted as one data sample. If, for example, the data are time series of various geophysical parameters, the first column of the matrix contains the sampling times while the following columns carry the n-1 parameters. Thus, each row of the matrix represents one data sample measured at the time given in the first component.

Most of the commands included in the EXTRACT menu (see Table EXTRACT) are used to set the conditions for extracting data from the input file to the data matrix. If a new data matrix was created, the mapping between the columns of the input file currently open and the columns of the data matrix must be defined. Reading the data according to this definition is done by one call to READ. If data from other files are to be stored into an existing data matrix, EXTRACT has to be left to open the new input file, and in a new call to EXTRACT the mapping and the reading is performed once again.

Creating a new data matrix (and deleting the existing one) is done by DIMENSION. DIMENSION has one parameter specifying the number of columns of the new data matrix. To read in e.g. three different time series, the dimension has to be at least four, - one column for the time axis and three data columns. If some additional columns are needed for results (e.g. arithmetic operations between the columns) the dimension must be large enough to accomodate these columns as well.

SHOW informs the user about the actual status of the data matrix. In particular the number of columns and the number of data samples are displayed. Additionally for each of the columns the mapping to

the input file, the constraints for reading the data, the minimum and maximum values as well as the mean value are shown.

MAP is currently not active. In the next version, the submenu MAP will be used to define the mapping of data into the data matrix, and MAP will replace the commands COMPONENTS and CONSTRAINTS.

The submenu LINKS is used to define the links between the data previously read in and data of the currently open input file. With links specified, data may be read in from the file and added to the existing samples in the matrix. The links define the column(s) of the data matrix, which are to be compared to specified column(s) in the data file. For example, it is thus possible to link two time series with different sampling times into one data matrix. The samples of the second series, which have a common time with one of the existing samples are added to the respective sample, while those not already present are added at the end of the data matrix. LINKS calls one of the standard editing menus for two-dimensional arrays (see Sect. 2.3.4). For each link, the number of the column in the data matrix and the number of the column in the data file have to be specified.

Currently, the command COMPONENTS defines the relation between the columns of the data matrix and the components of the input file. COMPONENTS has a varying but even number of parameters. The first of each parameter pair contains the number of the column of the data matrix, to which the new data is going to be written, while the second number denotes the component of the input file from which the data are read in.

CONSTRAINTS allows the user to define some constraints for reading data into a specified column of the data matrix. Therefore, the first parameter of the command is the index of the matrix column to which the constraints apply. In addition to this index several sets of constraints may be specified. Each of these has a first parameter denoting the index of the column of the input file to derive the constraints, and a second and third parameter defining the lower and the upper boundary of the constraints, respectively. A value is read to the column specified as the first parameter only if all values of the data file columns are between the lower and the upper boundary of the respective constraints. A sample may only be included in the data matrix if the first column satisfies its constraints. The number of constraints allowed for each column of the matrix depends on the main programme. The standard is three.

The commands STEP and OFFSET apply to equidistant time series only, with EXTRACT recognizing automatically if a file contains equidistant (time) series. STEP defines the step width used to extract data from the equidistant time series, while OFFSET sets the first value of the time series to be used.

READ actually transfers the data of the selected components from the input file to the data matrix according to the conditions defined by the commands explained above. READ has a switch, which specifies whether the data is to be added to already existing samples. If the switch is set to YES, the data are read in according to the links defined with LINKS. The default links are (1,1), i.e. the first column of the matrix is linked to the first column of the data file. It should be noted that merging two data files with the

```
> READ YES
```

command may be quite time consuming, since the search activities involved may be numerous.

The SORT command allows the user to sort the data samples of the matrix according to increasing values of the column specified as parameter of SORT.

Table 3.3:

TRANSF (V.2.0)	The TRANSF main menu
EDIT	Edit data (delete, multiply, offset,...)
COORDINATES	Perform coordinate transformations
LOG_EXP	Calculate logarithms or exponentials
CALCULUS	Perform task out of calculus (diff., integr.)
VECTOR_CALC	Vector calculus (norm, curvature,...)
CREATE	Create records or columns using functions or formulae
TESTS	Perform statistical tests on data
REGRESSION	Perform regression analysis
OUTPUT	Print or output the data

3.2.1   MAP: Map the data from external files into the data matrix

Not yet included

3.3   TRANSF: Transform the data vector

The TRANSFORM menu is used to change the content of the data matrix by editing and mathematical operations. The structure of the menu is given in Table TRANSF.

As usually, SHOW displays the current status of the data matrix. In particular, that is the actual number of columns, and for each of the columns the number of valid data (i.e. number of samples without missing values in the respective column), the arithmetic mean, the maximum and the minimum value.

All specific commands of TRANSF refer to submenus, which are described in the subsequent subsections.

3.3.1   EDIT: Edit the data matrix

The EDIT submenu allows to edit the structure of the data matrix and to perform simple arithmetic operations on the columns. The menu is given in Table EDITDT.

DELETE_COLUMN deletes a column from the data matrix. Besides the index of the column to be deleted, DELETE_COLUMN has an additional switch, defining whether the dimension of the data matrix shall be reduced after deleting the column. If the switch is set to YES, the columns behind the delete column are moved one index down, and the number of columns is reduced by one. If the switch is NO, the desired column is emptied, and all other columns remain unchanged.

REMOVE_GAPS removes all samples from the data matrix with a missing value in any of its columns. REMOVE_GAPS has no parameter.

MEAN subtracts from each desired column the mean values of this column. If no parameter string is given, MEAN acts on all columns. An example of this command is

Table 3.4:

EDITDT (V1.0)	Arithmetic operations on the data matrix
DELETE_COLUMN	Delete a column [reduce dimension Y/N]
REMOVE_GAPS	Remove rows containing missing values
MEAN	Subtract mean value from columns
MULTIPLY	Multiply data with a factor for each component
OFFSET	Subtract an offset for each component
ABSOLUT	Use absolute values [Y/N for each column]
INVERT	Compute the inverse value [Y/N for each column]

Table 3.5:

COORDI (V1.0)	Coordinate transformations
CARTHESIAN	Transform polar coordinates into carthesian
POLAR	Transform carthesian coordinates into polar

> mean 2,3,4

which removes the mean value from the columns 2, 3, and 4.

With MULTIPLY, the columns are multiplied by constant factors. Thus, MULTIPLY has n parameters, where n is the number of columns in the data matrix. The default for any factor is 1, which leaves a column unchanged. Thus,

> multiply 1.,,5.,,-3

will leave the columns 1,2 and 4 unchanged, while the third and fifth columns will be multiplied by 5 and -3, respectively. Any missing values are, of course, not changed.

OFFSET is used to subtract constant offsets from the data in different columns, i.e.

$$y_{ij} = y_{ij} - o_j \, , \tag{3.1}$$

where y_{ij} denotes the value in the i-th row and j-th column and o_j the offset defined for the j-th column. As is the case for MULTIPLY, OFFSET has n parameters, specifying the offset for each column. Default is 0, leaving a column unchanged.

ABSOLUTE also has n switches, specifying whether or not the data in the respective columns are to be replaced by the absolute values.

The syntax of INVERT is similar to ABSOLUTE and it is used to compute for each column as specified by the switches given the inverse values, i.e.

$$y_{ij} = frac1y_{ij}. \tag{3.2}$$

3.3.2   COORDI: Perform coordinate transformations

This submenu allows to convert data given in carthesian coordinates into polar coordinates and vice versa. Both commands have two parameters, indicating the columns of either amplitude and the angle, or the x- and y-coordinates.

Table 3.6:

LOGEXP (V1.0)	Calculate logarithmic or exponential functions
MINIMUM	Set minimum value to be used for logarithm
BASE	Set base for logarithm or exponential
ABSOLUT	Use absolute values [Y/N]
LOGARITHM	Calculate logarithm of all or given components
EXPONENTIAL	Calculate exponential of all or given components

Table 3.7:

CALCUL (V1.0)	Differentiate or integrate data columns
DIFFERENTIATE	Differentiate data with respect to 1st component
INTEGRATE	Integrate data with respect to first component
CUT_POLY	Cut out polynomial part by integration

3.3.3   LOG_EXP: Calculate the logarithm or exponential of columns

LOG_EXP allows to calculate the logarithm or exponential of selected columns. The original data is replaced by the results.

The three commands MINIMUM, BASE, and ABSOLUTE define conditions for the calculations. MINIMUM sets a lower boundary for calculating the logarithm of a data point. Values below this boundary are treated as missing values. With BASE, the base for the logarithm or exponential function is specified. Finally, ABSOLUTE sets a switch, which determines whether the original or the absolute values of the data are used in the calculations.

LOGARITHM or EXPONENTIAL perform the actual calculation. If these commands are used without parameter string, they affect all columns. If a parameter string is supplied, the action is restricted to only those columns indicated by the indices given in the parameter string.

3.3.4   CALCULUS: Perform simple calculus operations

CALCULUS performs some simple calculus operations on the data matrix. Currently, this menu has the structure given in Table CALCUL.

With the two commands DIFFERENTIATE and INTEGRATE it is possible to differentiate or integrate the data of the columns specified in the parameter string with respect to the first component of the data matrix. If no parameter string is supplied, all columns except the first one are acted on.

The CUT_POLY command is designed to cut out any polynomial part below the degree supplied as parameter by integration. This command eventually will supply a unique method for trend determination in time series with low frequency noise. However, currently the results are not satisfying.

For all three commands, the original data are replaced by the results of the calculation.

Table 3.8:

VECCAL (V1.0)	Perform simple operations of vector calculus
NORM	Calculate norm of data vectors [column index]
CURVATURE	Calculate curvature of data vectors [see info]
STRESS	Calculate stress components due to tangential friction

Table 3.9:

CREATE (V1.2)	Create the contents of the data matrix
RECORDS	Create workfile records from start to end with step
INTEGERS	Edit integer parameters of function
REALS	Edit real parameters of function
FUNCTION	Use user defined function to calculate new column
COLUMN	Calculate a new or overwrite an old column (in, i1, i2, op)

3.3.5 VECTOR_CALC: Perform simple vector calculus operations

VECTOR_CALC supplies some methods of vector calculus, which are helpful in studying the properties of vector quantities. In this menu, each row of the data matrix is interpreted as a sample of a parametrized vector of the dimension m=n-1, where n is the number of columns in the data matrix. The first column denotes the parameter, and in most cases this is the time. The menu has the structure given in Table VECCAL.

The norm of the vector is calculated by NORM. NORM has one parameter, specifying the column to which the resulting norm is written. Using DIFFERENTIATE (see Subsect. 3.3.4) and NORM together, both velocity and acceleration of a vector path may easily be calculated.

The direct formulas to calculate the curvature of a path involve several differentiations, which are numerically difficult to handle. Therefore, CURVATURE fits a m-dimensional pseudosphere to m adjacent samples, and the curvature is calculated as the inverse radius of this pseudosphere. Thus, in the case of three-dimensional vectors, a sphere is fitted to three adjacent points, and the curvature, the radius, and the coordinates of the centre are calculated. To use CURVATURE, the dimension of the data matrix must be n+5. The coordinates of the centre are written to the columns n+1, ..., n+3, the radius of the sphere to column n+4 and the curvature to n+5. CURVATURE has a switch, to decide whether or not the results are displayed during calculation.

3.3.6 CREATE: Create the contents of the data matrix by user defined functions

CREATE is a menu to create new (empty) records in a data matrix, fill the data matrix with synthetical values, or calculate new columns from existing data. The menu has the structure given in Table CREATE.

RECORDS creates an new data matrix, which has values only in the first column. Before applying RECORDS the matrix has to be defined by

Table 3.10:

TESTS (V1.0)	Do statistical tests on data
MANN_KENDALL	Calculate C-functions of Mann-Kendall rank test

```
> extract dim=x
```
were x denotes to the number of columns (see Sect. 3.2). The parameters of RECORD are the start value, the end value, and the step for the first column. Thus, the number of records created is determined by these parameters, too.

The command INTEGER, REAL and FUNCTION belong to a module, which supplies a function to be used for the calculation of synthetical data. INTEGER and REAL are used to edit any integer and real parameters, respectively, of the function. Both commands are calls to the respective EDITXX version. The mathematical function supplied depends on the main programme. If, e.g., the function is a sum of harmonic oscillations, then the integer parameters would be the number harmonic constituents and the degree of an additional polynomial, and the real parameters would be amplitude and phases of the constituents as well as the coefficients of the polynomial. If the functions is the pole tide due to polar motion, there would be no integer parameters, and the two real parameters would be longitude and latitude of the desired location.

With COLUMN, a hitherto empty column may be filled or an old column replaced with the results of an aritmethic operation between different columns. The parameter string of COLUMN must contain a valid equation (see sect. 2.1.1 for more details about equations). Column names are denoted by C_i, where i is the index of the column. Valid examples of COLUMN are

```
> column c5=c2+3*c3^ 0.5 - c3/c2,
> column c1=1/c1.
```

3.3.7 TESTS: Perform statistical tests on the data

In many cases, statistical tests are helpful in the assessment of data quality or the detection of a certain phenomena. Currently, the menu TESTS only supplies the Mann–Kendall C-Tests, described by Goosen and Berger (1986). Plots of the C-functions calculated according to the Mann-Kendall test are indicative for sudden shifts in a time series. Therefore, this test is a useful tool in the assessment of time series. TESTS currently has the structure given in Table TESTS.

MANN_KENDALL performs the test on the second column of the data matrix, while the two resulting C-functions are written to the third and fourth column of the matrix. Therefore, when using MANN_KENDALL, the matrix must have a dimension of at least four.

3.3.8 REGRESSION: Perform regression analyses between the data columns

Regression analyses are an useful tool in studying the relation between different physical quantities. The current version of REGANA allows for simple univariant regression analyses. The menu has the structure given in Table REGANA.

Table 3.11:

REGANA (V1.0)	Perform linear regression analyses
COLUMN	Edit columns to be used
REGRESS	Calculate regression line
RESIDUAL	Create regression line and residuals
LOAD	Load results into text buffer

Table 3.12:

OUTPUT (V1.0)	Output of data
PRINT	Print data

COLUMN defines the indices of the columns of the data matrix which are used as input data and to store the results of the regression analyses. COLUMN calls the EDITIA menu described in Sect. 2.3.2. The dimension of the index array may be 2, 4, or 6. The first two elements of the index array denote the columns selected for the analysis. The indices 3 and 4 specify the columns selected for the results and the residuals of the regression considering the data column specified by the second index to be a function of the one specified by the first index. The indices 5 and 6 specify those columns used for results and residuals, respectively, of a regression considering the first column to be a function of the second one.

REGRESS calculates the two regression lines, and the result is displayed. Afterwards, RESIDUAL may be used to calculate the residuals and store them into the data matrix according to the definition of COLUMN. Finally, LOAD allows to load the regression and correlation coefficients into the standard text buffer, which is e.g. available for an output in a plot.

3.3.9 OUTPUT: Write the data matrix to the programme's list file

With OUTPUT, the contents of the (transformed) data matrix may be included on the list file. Currently, the Output has no commands for the layout of the listing.

With PRINT, the contents of the data matrix are written to the list file. The format of the data is fixed.

3.4 WRITE: Menu to output data and/or results

Depending on the tasks of the programme, the WRITE menu may contain several commands to output different datasets or results to independent disk files, or to select between various kinds of formats for the output. The WRITE menu has the form given in Table WRITE.

WRITE does not offer any facilities for file opening tasks. These have to be performed within the FILES menu (see Sect. 2.3).

Table 3.13:

WRITE (V1.0)	Write data to external files
Task1	Write data to file 1
...	...

Chapter 4

PLOTSOFT: a bit more than a graphical kernal system

Part II

RHEOLOGY

Chapter 5

URANIA: Editing the model of the EARTH or any other celestial body

Chapter 6

SYSPONSE: Calculating the System's Response to External Forcing

SYSPONSE is a programme for the computation of the response function of a non-relativistic, spherically symmetric, non-rotating, viscoelastic, isotropic, selfgravitating model of a celestial or planetary body. Most frequently the body considered will be the Earth. For a harmonic body forcing, such as the tides, the response function is calculated for a harmonic excitation of an unit amplitude. Loading response functions are computed for a space-time unit impulse. For convention, the more general term 'structural model' instead of the special term 'Earth model' will be used.

SYSPONSE supplies facilities to address different fields of impulse deformations of a structural model. These fields are:

- deformations due to body forcing, for instance by the tidal forces of other celestial bodies,

- deformations due to a surface loading,

- free oscillations of the considered structural model.

The fundamental outputs of SYSPONSE are the Love-Shida numbers (both for body and loading forces), Molodensky's auxiliary solutions, and eigenfrequencies well as eigenfunctions. All output parameters depend upon the spherical harmonic degree and the Love-Shida numbers and Molodensky's auxiliary solutions as well upon frequency and radius. The body tide Love-Shida numbers and Molodensky's auxiliary solutions constitute the input parameters for the programme PERTURB where they represent the undisturbed solution.

The next table shows the available commands and submenus of SYSPONSE.

In the subsequent sections descriptions of the following menus are given: FILES, DEFINE, F_DEFINE, SELECT_MODEL, VIEW and WRITE.

Of the remaining six specific commands of SYSPONSE five will start calculations. By SINGLE the computations of Love-Shida numbers and/or Molodensky's auxiliary solutions is started. COMPLETE initiates the calculation of the complete set of output parameters required for further processing by the programme PERTURB. After the RELAXATION command the relaxation of the viscoelastic moduli used in the structural model is determined. STRESS will start the calculation of the

Table 6.1:

SYSPONSE (V1.2)	Calculating the response functions
FILES	Open files for I/O
DEFINE	Define parameters
SELECT_MODEL	Select extent of theoretical model
SINGLE	Calculate desired Love numbers, show, print [Y/N]
COMPLETE	Calculate complete set of parameters for perturbation meth.
RELAXATION	Calculate relaxation of moduli
F_DEFINE	Define parameters for free mode determination
STRESS	Calculate the surface stress function
FREE_MODES	Determine free modes
VIEW	View and/or print results and other parameters
WRITE	Write results to output file
RECOVER	Try to recover last results

Table 6.2:

FILES (V1.0)	Opening the input and output files
MODEL	Open input file for structural model
LOVE_NUMBERS	Open output file for Love numbers
FREE_MODES	Open output file for free modes
PARAMETRES	Open output file for parameters

surface stress function. Finally, the FREE_MODES command will initiate the determination of free modes.

Recover is a command which will recover the last results obtained by a SYSPONSE run. In such cases where the programme is aborted due to cpu-time limit RECOVER helps to access the results calculated prior to abortion.

6.1 FILES: Open input and output files for SYSPONSE

There are four different files to be opened in FILES. The appropriate commands are given in the table.

MODEL opens a new input file of the structural model and reads in the data contained in the file. If, e.g., the input file name is 'earth3.dat' opening and reading in is achieved by

> MODEL earth3.dat

LOVE_NUMBERS opens an output file for the resulting Love-Shida numbers and auxiliary solutions after Molodensky. An output file for the results of a free mode determination is opened by FREE_MODES. By PARAMETER an output file is opened for parameters of the structural model possibly required subsequently by other programmes such as GREENF.

Table 6.3:

SELECT (V1.3)	Selecting the extent of the model
DYNAMIC	Calculate Love numbers for dynamical case
TOROIDAL	Molodensky's auxilary solution for toroidal
SPHEROIDAL	Molodensky's auxilary solution for spheroidal
BODY_TIDES	Calculate Love numbers for body tides
LOADING_CASE	Calculate Load Love numbers
TRACTION	Calculate Love numbers for surface traction
VISCO_ELASTIC	Calculate visco-elastic Love numbers
LAPLACE	Use Laplace instead of Fourier transform
FLUID	Consider planet as completely fluid

6.2 SELECT_MODEL: Select extend of model

In SELECT_MODEL the extent of the theoretical model standing behind all types of calculations carried out by SYSPONSE is determined. As well, it is specified to calculate Love-Shida numbers and/or Molodensky's auxiliary solutions.

The next table displays the commands available in SELECT_MODEL.

The DYNAMIC switch is related to the forces of inertia which may be taken into account in the computations. Applying the DYNAMIC switch will lead to a dynamic calculation whereas discarding DYNAMIC will give rise to a static treatment of the response problem.

Molodensky's auxiliary solutions exist for the toroidal and the spheroidal case which are activated by the respective switces TOROIDAL and SPHEROIDAL to achieve the corresponding computations.

The BODY_TIDES switch is used to select body tide Love-Shida numbers for a computation. The LOADING_CASE switch serves for the selection of load Love-Shida numbers, while setting the TRACTION switch results in the calculation of Love numbers for surface traction. In the current version, only one of these switches may be set at a time.

SYSPONSE is able to conduct the calculations either for a purely elastic or a viscoelastic structural model. To skip between these possibilities the VISCO_ELASTIC switch is used.

By the LAPLACE switch a skipping between calculations in the Laplace and Fourier domain is possible. The Laplace domain should be used if aperiodic phenomena are of interest. Periodic phenomena should be treated in the Fourier domain.

The planetary or Earth model may be considered as completely fluid (the shear modulus equals zero everywhere) by the FLUID switch.

6.3 DEFINE: Define parameters for the calculation of Love Numbers

DEFINE serves for the definition of the parameters which SYSPONSE will use in its calculations performed by the SINGLE command. The available commands are displayed in the following table.

Table 6.4:

DEFINE (V1.1)	Defining the calculation parameters
F_INTERVAL	Set frequency interval (enter log.)
DF	Set step for frequency (enter log.)
RADIUS	Edit radius values
DEGREES	Edit degrees of spherical harmonics wanted
ACCURACY	Enter accuracy of integration
SHEAR_MOD	Enter minimum for relaxed relative shear modulus
SURFACE	Only the surface Love numbers are stored
DISPLACE	Output displacements instead of Love numbers

In general, Love-Shida numbers and Molodensky's auxiliary solutions depend upon frequency (however, not for static and purely elastic computations), radius, and harmonic degree. Thus, the desired values of these parameters have to be defined for the calculations.

The frequencies are defined by the commands F_DEFINE and DF.F_INTERVAL is used to specify the frequency or Laplace parameter interval.The F_INTERVAL command is followed by two real numbers seperated by a blank; the first number gives the lower bound and the second number the upper bound of the interval. The bounds are enterd as the decadal logarithms of frequency or the Laplace parameter in units of 1/second. The use is illustrated by an example:

> `F_INTERVAL -4.0,1.0`

which means that an interval ranging from 0.0001 to 1.0 1/second has been defined. The step length for the sampling of the specified interval is entered by the DF command followed by a single real number giving the decadal logarithm of the step length in units of 1/second as shown in the next example:

> `DF 0.5`

will generate a frequency sampling with logarithmic steps of 0.5. Regarding both examples as a whole the so defined frequencies or Laplace parameters are (as decadal logarithms) -4.0, -3.5, -3.0, ..., 0.0, 0.5, and 1.0.

The radii for which the Love-Shida numbers or Molodensky's auxiliary solutions will be calculated, are edited in the RADIUS submenu as a one-dimensional array of values of radii. A general description of such kind of menu is given in Sect. 2.3.2. It is important to note, that the values of radii have to be entered in units of kilometres.

DEGREES serves for editing a one-dimensional integer array of harmonic degrees for which the Love-Shida numbers or Molodensky's auxiliary solutions will be computed. A general description of this kind of menu is supplied in Sect. 2.3.2.

The Love-Shida numbers or Molodensky's auxiliary solutions will be determined by numerical integration. A relative accuracy for the integration is defined by the ACCURACY command followed by a real number. For instance,

> `ACCURACY 0.001`

results in an integration accuracy of the 0.1 % level.

The SHEAR_MOD command followed by a real number defines an absolute minimum value of the viscoelastically relaxed shear modulus. If in the course of the calculations a relaxed value of a shear

Table 6.5:

DEFINE (V1.0)	Defining the free modes calculation parameters
INTERVAL	Set time interval
STEP	Set time step
ACCURACY	Set accuracy in frequency
DEGREES	Set degrees of wanted mode
OFFSETS	Set offsets of starting overtone
OVERTONES	Set number of overtones

modulus at any depth will fall below this minimum, this particular value will automatically be set zero. This minimum value of the shear modulus has to be given in units of Pascals. For example,

> SHEAR_MOD 5.5E+3

will define a minimum value of the shear modulus of 5500 Pa.

The SURFACE switch is used to specify whether the computed Love-Shida numbers are only stored as values at the surface of the structural model or kept for all defined radii.

For some applications it may be very useful to generate an output in terms of actual displacements instead of Love-Shida numbers. This kind of output is activated or deactivated by the DISPLACE switch.

6.4 F_DEFINE: Define parameters for the computation of free modes

F_DEFINE provides the necessary commands to define the appropriate parameters for the free modes determination. All commands are displayed in the next table.

INTERVAL is used to define the period interval in which free modes are searched for. INTERVAL has two parameters, the first giving the lower bound and the second number defining the upper bound of the period interval in units of seconds.

The resolution of the free mode search is specified by the STEP command, where the step is given in seconds, too. The step should by selected small enough to separate all neighbouring free modes. However, a step smaller than required will increase the required computing time.

ACCURACY defines the accuracy for the determination of the frequency of a free oscillation. The real number following the ACCURACY command must be given in units of 1/second.

DEGREES invokes the EDITIA menu for editing the spherical harmonic degrees of the desired free oscillation.

For some spherical degrees, some modes don't exist, and the first existing mode is already an overtone. To account for this, for each spherical degree an offset may be specified, which is done by OFFSET. OFFSET is a call to the EDITIA menu.

OVERTONES specifies the number of overtones to be determined for each harmonic degree.

Table 6.6:

VIEW (V1.0)	Viewing parameters and ouptut
EARTH_MODEL	Show complete earth model
LOVE_NUMBERS	Show Love numbers (h,l,k [Y/N])
FREQUENCY	Select frequency interval to be shown
RADIUS	Select radius interval to be shown
DEGREES	Select interval of degrees to be shown
LIST_FILE	Set list file output to [Y/N]
SCREEN	Set screen output to [Y/N]

6.5   VIEW: View Love numbers and free modes

A general outline of VIEW is given in Sect. **??**. Here, only those commands specific for SYSPONSE are explained. The commands of VIEW are shown in the table below.

The FREQUENCY, RADIUS and DEGREE commands are used to select the amount of data displayed. FREQUENCY followed by two real numbers giving the lower and upper bounds of the frequency or Laplace parameter interval defines an according interval of Love-Shida numbers to be viewed or written. The real numbers must be entered as decadal logarithmic values of the frequency or Laplace parameter in units of 1/second. For instance,

> FREQUENCY -1.0,2.0

will define a viewing those Love-Shida numbers which have been computed in the interval from 0.1 to 10 s^{-1}

An interval of radii is constrained by the RADIUS command followed by two real numbers for the lower and upper bound of the interval. The numbers must be given in units of kilometres. A viewing of all results computed for the radius interval from 3500 km to 6220 km, for example, is achieved by

> RADIUS 3500.0,6220.0

DEGREES followed by two integers, the lower and upper harmonic degree to be viewed, defines an interval of harmonic degrees of results to be extracted in the according menu. For example,

> DEGREES 500,1270

will restrict the results to harmonic degrees from 500 to 1270.

EARTH_MODEL is used to view the complete structural model presently in use. The LOVE_NUMBER command serves for the selection of any combination of the three Love-Shida numbers in the sequence h, l, and k. LOVE_NUMBER has to be followed by three characters (either Y or N) separated by commas. For instance, the command

> LOVE_NUMBERS Y,N,Y

will only prepare Love numbers h and k for viewing and omit Shida numbers l.

Table 6.7:

WRITE (V1.0)	Writing the output
SEQUENTIAL	Write sequential file
INERTIA	Write file for INERTIA in GREENF
COMPLETE	Write file for PERTURB
STRESS	Write stress function
ROOT	Write roots
EIGENFUNCTION	Write eigenfunction
FREQUENCY	Select frequency interval to be written
RADIUS	Select radius interval to be written
DEGREES	Select interval of degrees to be written

6.6   WRITE: Write Love Numbers or free modes to files

A general outline of WRITE is given in Sect. 3.4. Here, only those commands specific to SYSPONSE are explained. The commands of the WRITE menu are shown in the tables below.

The commands FREQUENCY, RADIUS and DEGREE serve for the selection of the amount of data to be written. These commands are fully described in the previous section (6.5).

WRITE provides five additional specific commands. With SEQUENTIAL, the currently available Love numbers are written to a standard data file. INERTIA creates an advanced segmented file containing Load Love numbers, Body Tide Love number and Earth model parameters as required for the calculation of the Green's functions for changes in the moment of inertia. COMPLETE creates an advanced segmented file containing Body Tide or Load Love Numbers, the Auxiliary Solutions and model parameters for PERTURB.

STRESS creates a file with the surface stresses available. ROOT and EIGENFUNCTION write the free mode periods and their eigenfunction, respectively to external files.

Chapter 7

PERTURB: Perturbations of spherical symmetry and elasticity

PERTURB is a programme for the computation of first-order perturbation effects on surface Love-Shida numbers, tilt and normal gravity factor for the response on the harmonic body tides forcing of a structural model of a non-relativistic celestial or planetary body. This body in most applications probably will be the Earth.

PERTURB addresses a number of deformational perturbation effects that are due to first-order deviations from a static SNREI structural model (SNREI= Spherically symmetric, Non-Rotating, Elastic, Isotropic). Namely, PERTURB is designed for the treatment of the following effects:

- effect of the forces of inertia, that is a dynamical instead of a static treatment,

- effect of the rotation and ellipticity of the structural model by taking into account the centrifugal and the Coriolis forces,

- effect of viscoelasticity of shear modulus and Lamé parameter,

- viscoelastic amplification effect due to the amplification of viscoelastic anomalies when looking at them at various frequencies,

- effect of lateral heterogeneities of shear modulus and Lamé parameter,

- effect of boundary layer undulations, that is deviations from the spherical symmetry of boundaries of the structural model (in preparation),

- effect of anisotropy of the shear modulus (in preparation).

It has to be noted that some of the mentioned effects may alternatively be computed by the SYSPONSE programme. These are the effects of inertia and viscoelasticity. However, one has to be aware of the different methods of computation: PERTURB is exclusively based upon the first-order perturbation approach! In particular, SYSPONSE will give a better accuracy of computation for these effects! Especially the viscoelastic frequency-dependence obtained for low frequencies by PERTURB will differ significantly from what is obtained by SYSPONSE.

Table 7.1:

PERTURB (V1.0)	Calculation of perturbations Love Numbers
FILES	Open files for I/O
ED_ASPHERIC	Menu for editing aspherical earth model structure
DEFINE	Define parameters for calculations
SELECT	Select functions to be calculated
CALCULATE	Calculate desired functions, print, write [Y/N]
VIEW	View and/or print results and other parameters
WRITE	Write results to output file

Table 7.2:

FILES (V1.0)	Opening of input/output files
START_MODEL	Open input file for spherical start earth model parameters
ASPHERICAL	Open input file for aspherical earth model structure
RESULTS	Open output file for results

The output generated by PERTURB consists in a set of perturbed Love-Shida numbers, a complete set of all Love-Shida numbers required as input for the programme GeoTIDES, and a file containing the aspherical structure of the structural model that is used.

The available commands of PERTURB are shown in the table below.

7.1   FILES: Opening input and output files for PERTURB

The FILES menu of PERTURB serves for the opening of three files. The commands are shown in the table below.

START_MODEL opens the fundamental input file containing the purely spherical structure of the model to be used. ASPHERICAL is used to open an output file for the parameters describing the aspherical structure. RESULTS opens the output file for the results of the computations.

7.2   ED_ASPHERIC: Edit the aspherical structure of the model

ED_ASPHERIC supplies three submenus for the editing of the aspherical structure of the model for which the perturbations will be computed. Commonly, the aspherical structure is read in from a particular input file. If, however, the user wants to make according changes, these are achieved in the ED_ASPHERIC menu. The table shows the commands and submenus.

Presently, only the LATERAL_HET submenu is available for. LATERAL_HET serves for the editing of the lateral heterogeneities of the seismic P- or S-wave velocities.

Table 7.3:

ED_ASPH (V1.0)	Editing the aspherical structure
LATERAL_HET	Menu for editing lateral heterogeneities
BOUND_UNDUL	Menu for editing boundary layer undulations (in prep.)
ANISOTROPY	Menu for editing anisotropic properties (in prep.)

Table 7.4:

LATHET (V1.0)	Editing lateral heterogeneities
NUMBER	Enter total number of laterally heterogeneous layers
LAYER	Enter index of lat. het. layer and edit layer

7.2.1 LATHET: Editing the lateral heterogeneities

The commands available in LATHET are displayed in the next table.

The NUMBER command followed by an integer is used to define the total number of laterally heterogeneous layers which will be introduced into the structural model. The LAYER command followed by an integer giving the layer number invokes the LAYER edit menu for the heterogeneity of that specific layer.

LAYER: Edit the lateral heterogeneity within a layer

In LAYER the heterogeneous layers and their properties are defined. The present models of heterogeneities of the Earth's mantle supply the related information in terms of either seismic P- or S-wave velocities for a certain range of radius. The deviations of the seismic velocities from the spherical symmetric structural model depend on radius, geographical colatitude and longitude. Thus, the deviations are commonly expanded in terms of series which encompass products of purely radial-dependent polynomials (commonly Legendre polynomials) and associated Legendre polynomials for the dependence on colatitude and longitude. The associated Legendre polynomials introduce longitude dependent cosine and sine terms to which specific expansion coefficients are attributed. These expansion coefficients, however, at the same time account for the radial expansion, so that in effect the coefficients carry three indices, one radial index giving the radial polynomial degree and two indices for the involved angles actually giving the harmonic degree and order of the associated Legendre polynomial, respectively.

LAYER supplies the commands shown in the next table.

By the LO_UP_RAD command the radial extension of the heterogeneous layer is defined. LO_UP_RAD is followed by two real values separated by a comma; the first value gives the lower radius, the second the upper radius.

The HARM_DEG command is used to define the maximum harmonic degree appearing for the associated Legendre polynomials. HARM_DEF is followed by an integer giving the maximum degree.

RADIAL_DEG with an integer following, is used to enter the maximum degree occurring in the radial polynomials.

Table 7.5:

LAYER (V1.0)	Editing a lateral heterogeneous layer
LO_UP_RAD	Enter lower and upper radius of heterog. layer in km
HARM_DEG	Enter max. harmonic degree and order of perturbation
RADIAL_DEG	Enter max. degree of radial polynomials of perturbation
VP	Switch for heterogeneity given as P- or S-wave velocities
POWER	Enter power for which P/S-wave velocity is expanded
UNITS	Conversion factor of coefficient units to cgs-units
COEFFICIENTS	Menu for editing the expansion coefficients of het'geneity

Table 7.6:

COEFF (V1.0)	Editing the coefficients of heterogeneities
R_DEGREE	Enter radial polynomial degree of coefficient
H_DEGREE	Enter spherical harmonic degree of coefficient
H_ORDER	Enter spherical harmonic order of coefficient
COS_COEFF	Enter cosine associated harmonic coefficient
SIN_COEFF	Enter sine associated harmonic coefficient

VP is switch between the P- and S-wave velocities as the fundamental information for the heterogeneity. Either a heterogeneous layer is defined in terms of P-wave or S-wave velocities.

By the POWER command followed by an integer the power of the deviations of P- or S-wave velocities is entered. For instance, a 1 is entered if the heterogeneities are given as simple deviations of wave velocities; a 2 is entered if the heterogeneities are given as squared deviations of wave velocities.

UNITS is followed by a real number that defines the conversion factor from the units used for the expansion coefficients to cgs-units needed in PERTURB.

The command COEFFICIENTS invokes a menu for editing the complete set of expansion coefficients of the current layer.

COEFFICIENTS: Edit the spherical harmonic coefficients of heterogeneities

All available commands of the COEFFICIENTS menu are shown in the next table.

As was explained earlier, the expansion coefficients of the seismic velocity deviations are related to a sine and a cosine term and carry three indices: one index for the degree of the radial polynomial, another for the spherical harmonic degree, and a last one for the spherical harmonic order.

R_DEGREE, H_DEGREE, and H_ORDER are each followed by an integer defining the radial polynomial degree, the spherical harmonic degree, and the spherical harmonic order of the coefficient, respectively.

COS_COEFF and SIN_COEFF are each followed by a real number defining the cosine and sine coefficients, respectively, of the specified radial degree, harmonic degree and harmonic order.

Table 7.7:

SELECT (V1.0)	Selecting the effects of perturbation
DYNAMIC	Include dynamic effects
ROT_ELL	Include effects of rotation and ellipticity
L_HETEROG	Include lateral heterogeneities
RELAXATION	Include effect of viscoelastic relaxation
AMPLIFICATION	Include effect of viscoelastic amplification

7.3   SELECT: Select extend of perturbations

The SELECT menu provides a set of switches for the selection of the extent of the theoretical model that will stand behind the computation of the perturbation effects on the Love-Shida numbers. The table gives a complete overview of the menu.

The DYNAMIC switch is used to select between the possibilities of a dynamical treatment of the problem, that is the accounting of the forces of inertia, and a purely statical treatment.

The ROT_ELL switch is related to the effects of rotation and ellipticity. Both effects are considered together because a rotating structural model necessarily is an elliptical one.

The L_HETEROG switch aims at the use of lateral heterogeneities that is continuous spatial variations of the elastic moduli at a given radius.

RELAXATION is a switch for the effect of a viscoelastic relaxation of the moduli. Warning: the results obtained for a pronounced relaxation as occurring for rather low frequencies of forcing may considerably differ from what comes out of a exact treatment of viscoelasticity!

The AMPLIFICATION switch is used for the effect of the so-called viscoelastic amplification effect. This effect is related to the viscoelastic treatment of lateral heterogeneities. The heterogeneities have been observed for a certain reference frequency. When looking at frequencies different from the reference frequency the magnitude of heterogeneity will be different from the reference magnitude of heterogeneity due to the effect of frequency-dependence of the moduli. In the present theory the viscoelastic amplification effect is supposed to be solely induced by temperature heterogeneities. Chemical heterogeneities, for example, are disregarded.

7.4   DEFINE: Define parameters for the computation of perturbations

The definition of specific computational parameters is carried out in the DEFINE menu. The menu is shown in the next table.

In PERTURB the effects of perturbations in the case of body forcing are computed. The body forcing is always related to a set of spherical harmonic degrees and orders. In the present version, PERTURB allows for a computation of only one specific combination of a harmonic degree and order. This combination is defined by the B_DEGREE and B_ORDER commands which are followed by an integer that gives the harmonic degree and order, respectively.

Table 7.8:

DEFINE (V1.0)	Defining computational parameters
B_DEGREE	Enter harmonic degree of body forcing
B_ORDER	Enter harmonic order of body forcing
PERIODS	Enter interval of forcing period
DP	Enter period step
P_DEGREES	Edit harmonic degrees to be calculated
START_RADIUS	Enter start-radius for num. integration
LOD	Enter length of day in seconds

Table 7.9:

VIEW (V1.0)	Viewing parameters and output
SNREI	Show parameters of the undisturbed structural model
PERTURB	Show parameters of current perturbations
RESULTS	Show the Love-Shida number perturbations

The body forcing is associated with forcing periods which fall into a certain interval. PERIODS followed by two real numbers separated by a comma specifies the interval of period which must be given in units of seconds. DP followed by a single real number specifies the step length by which the interval is sampled.

P_DEGREES invokes the common edit menu EDITRA for the harmonic degrees (the perturbations are expanded in terms associated spherical harmonics) for which the results have to be computed. A general description of the edit menu, which is used for one-dimensional arrays, is found in Sect. 2.3.2. The according harmonic orders ranging from the negative value of the degrees up to the positive value are defined, automatically.

LOD is followed by a single real number giving the length of day in units of seconds. Thus, by LOD the angular velocity of the rotation is defined.

7.5 VIEW: View the results of PERTURB

General information on the VIEW menus are available in Chapter **??**. The commands of the VIEW menu are displayed in the next table.

SNREI is used for the viewing of the parameters of the undisturbed structural model. PERTURB shows the set of perturbational parameters which are currently in used. RESULTS shows the computed perturbations of the Love-Shida numbers.

7.6 WRITE: Write the results of PERTURB to output files

General information on the WRITE menu are available in Chapter 3.4.

Table 7.10:

WRITE (V1.0)	Writing of output files
P_LOVE	Write Love-Shida number perturbations to a file
GT_FILE	Write a file containing all Love-Shida numbers for GeoTIDES
ASPHERICAL	Write aspherical structure to a file

The available commands of the WRITE menu are given in the next table:

P_LOVE writes the computational output of only the Love-Shida number perturbations to the according output file.

GT_FILE writes all Love-Shida numbers, that is the static elastic numbers, their viscoelastic increments, and the perturbed Love-Shida numbers, to an output file that will serve as an input for subsequent computations by the programme GeoTIDES.

ASPHERICAL writes the current aspherical structure of the model to the according output file.

Chapter 8

GREENF: Calculating Green's functions

GREENF calculates the Green's function for external periodic or aperiod loading on a viscoelastic Earth (or any body). The main menu of GREENF is given in the next table.

Currently, the chapter on GREENF is not finished. Only the menus are given but no explanations included.

Table 8.1:

GREENF (V4.1)	The GREENF main menu
FILES	Open files for I/O
DEFINE	Define parameters
SELECT	Select Green's Functions to be calculated
SINGLE	Inverse LT of a single degree
TEMPORAL	Transform all degrees into time domain
GREEN	Calculate desired Green's Functions
INERTIA	Determine Green's Function for moment of inertia
VIEW	View and/or print results and other parameters
WRITE	Write results to output file

Table 8.2:

FILES (V1.3)	The File menu
LOVE_NUMBERS	Open input file with Love numbers
GREEN_FUNCTIO	Open output file for Green fcts.
FIT_LAP	Open output for results of fitting in Laplace domain
FIT_K	Open output for results of fitting in wave number domain
LAPLACE	Open output file for spectra or inverse of LT
INERTIA	Open output file for the Eigenvalues of the Inertia tensor

Table 8.3:

DEFINE (V1.1)	Define parameters for Green's functions
D_INTERVAL	Set distance interval (enter log. of degrees)
DD	Set step for distance (enter log. of degrees)
R_INTERVAL	Set interval of radii
T_INTERVAL	Set interval of times
DEGREE	Enter maximum degree for expansion of fn
START	Enter start index for approximation
ACCURACY	Enter accuracy of integration
WEIGHT	Choose kind of weighting

8.1   FILES: Open input and output files for GREENF

8.2   DEFINE: Define parameters of the Green's functions

8.3   SELECT: Select Green's functions to be calculated

8.4   SINGLE: Calculate inverse Laplace Transform of a single degree

8.5   LAPINV: Calculate inverse Laplace Transforms

8.5.1   DEFLAP: Define parameters for the inverse Laplace Transform

8.5.2   LAPKIND: Select kind of base function of spectrum

8.5.3   DEFPEAK: Define peaks for variance spectrum of LT

8.5.4   LAPVIEW: View intermediate and final results of inverse Laplace Transform

Table 8.5:

SINGLE	Inverse LT of a single harmonic degree
DEGREE	Enter spherical degree
RADIUS	Enter radius to be used
LOAD	Load Love numbers of DEGREE and RADIUS
FUNCTION	Select function to be used
LAPLACE	Call submenu for Laplace Transform

Table 8.6:

LAPINV (V1.2)	Inverse Laplace Transforms
DEFINE	Define spectrum and/or inverse Laplace Transform
P_FITTINGS	Enter print extend for output of fittings
SPECTRUM	Calculate variance spectrum
PEAKS	Define peak parameters incl. number, size and form
ANALYSE	Analyse variance spectrum to find peak positions
ADJUST	Adjust peak positions
INVERSE	Calculate inverse Laplace transform
VIEW	View and print results
WRITE	Write results to output files

Table 8.7:

DEFLAP (V1.0)	Define parameters for the inverse LT
FREQUENCY	Enter frequency interval for spectrum
DF	Enter frequency step for spectrum
POLES	Set automatic frequencies for pole search
BASE_FUNCTION	Select base function for spectrum
INV_TIME	Enter time interval for inverse Laplace Transform
DT	Enter time step for inverse Laplace Transform
KNOWN	Set known frequencies
RECIPRO	Use the inverse of the Laplace Transform
P_FITTINGS	Enter print extend for output of fittings
P_ADJUST	Enter print extend for adjusting
P_MODEL	Enter print extend for model

Table 8.8:

LAPKIND (1.0)	Kind of base function
NONE	Use development into exponentials (1/(s+sv))
DELTA	Include constant term corresponding to delta
COMPLETE	Include constant term and 1/s

Table 8.9:

DEFPEAK (V1.0)	Define peak parameters
NUMBER	Enter maximum number of peaks collected
LEVEL	Enter minimum level of peak amplitude (var) to be used
SAMPLE	Sample peak definition
ACCURACY	Enter relative accuracy for adjusting
STEP	Enter log of step for adjusting
LOWER_BOUND	Lower bound for all parameters
UPPER_BOUND	Upper bound for all parameters
ABSOLUTE	Use absolute peak size instead of relative

Table 8.10:

LAPVIEW (V1.0)	View inverse LT
SPECTRUM	Show Laplace spectrum
PEAKS	Show peak parameters for Laplace spectrum
MODEL	Show model parameters
INVERSE	Show inverse Laplace Transform
LIST_FILE	Set list file output to [Y/N]
SCREEN	Set screen output to [Y/N]

Table 8.11:

LAPWRIT (V1.0)	Write inverse LT
SPECTRUM	Write current Laplace spectrum
PEAKS	Write current peaks
MODEL	Write current model parameters
INVERSE	Write current inverse Laplace Transform

Table 8.12:

VIEW	View Green's function
GREEN_FCT	Show Green function
DISTANCE	Select distance interval to be shown
RADII	Select radius interval to be shown
TIMES	Select interval of time to be shown
LIST_FILE	Set list file output to [Y/N]
SCREEN	Set screen output to [Y/N]

Table 8.13:

WRITE	Write Green's function
SEQUENTIAL	Write sequential file
GYROS	Write file for GYROS
DISTANCE	Select distance interval to be written
RADII	Select radius interval to be written
TIMES	Select time interval to be written

Part III

EXTERN: Modelling exogenic deformations

Chapter 9

GEOTIDES: Tidal Deformations and Perturbations of the Potential

GEOTIDES is a programme for the calculation of quantities related to the tidal deformations of a structural model which in most applications will be the Earth. GEOTIDES is able to calculate the surficial tidal displacements, the surface tilts, the surface normal gravity, the tide generating potential and acceleration which is primarily raised by the Moon and the Sun, and the so-called secondary tidal potential and acceleration, that is the additional gravitational potential and acceleration due to changes of the configuration of the Earth's mass by the tidal deformations themselves. The tidal quantities are determined for an viscoelastic aspherical structural model.

The basic input of GEOTIDES consists of a harmonic model of the tide generating potential and a set of Love-Shida numbers for an viscoelastic aspherical structural model. From this information the tidal quantities are determined essentially by appropriate summation of tidal potential terms and Love-Shida numbers and their perturbations.

In the present state GEOTIDES is able to treat the body tide problem.

The available submenus and commands of the main menu of GEOTIDES are given in the table.

Table 9.1:

GEOTIDES (V1.1)	Calculating the response of the Earth to tides
FILES	Open files for I/O
SELECT	Select Earth model properties to be used
SET_TIME	Set epoch interval, time step, time units
DEFINE	Define parameters for calculations
CALCULATE	Calculate desired output
VIEW	View and/or print results and other parameters
WRITE	Write results to output file with standard format
WR_GNUPLOT	Write results to output file with Gnuplot format

Table 9.2:

FILES (V1.0)	Opening input and output files
BODY_TID_POT	Open input file for tidal potential
LOVE_NUMBERS	Open input file for Love-Shida numbers
RESULTS	Open output file for results

Table 9.3:

Menu SELECT (V1.1)	Selecting the extent of the model
DYNAMIC	Use dynamic Earth model properties instead of static ones
ROT_ELL	Use the properties of a rotating and elliptical Earth model
HETEROG	Use the properties of a laterally heterogeneous Earth model

9.1 FILES: Open input and output files for GEOTIDES

FILES serves to open three files by the commands given in the next table.

BODY_TID_POT opens an input file for the harmonic model of the tide generating potential. Presently, there are three up-to-date models which are available. These are the models of Buellesfeld (1985), Xi (1987), and Tamura (1987) which are kept in the files buelles.pot, xi.pot, and tamura.pot, respectively. The most compact among the models is that of Buellesfeld with 665 tidal constituents (or equvalently, partial tides). Tamura's model comprises 1200 and Xi's model 3070 tidal constituents and yield a higher accuracy than does Buellesfeld's model, however they are much more time consuming. Therefore, the Buellesfeld model is recommendable for most applications.

LOVE_NUMBERS opens an input file for a complete set of Love-Shida numbers that are required for taking into account the effects of elasticity, viscoelasticity, effects of rotation and ellipticity, and effects of lateral heterogeneities. This set of Love-Shida numbers has been computed by the programmes SYSPONSE and PERTURB and in the final version been generated by the programme PERTURB.

RESULTS is used to open an output file for the results of the calculations of the present programme.

9.2 SELECT: Select extend of model

SELECT provides a set of switches for the selection of the extent of the theoretical model that will stand behind the computation of the tide related quantities. The table gives an overview of the specific commands.

The DYNAMIC switch is used to select the dynamic Love-Shida numbers. If DYNAMIC is set to 'no' only the static part of the active Love-Shida numbers will be in effect.

ROT_ELL is a switch related to the use of the effects arising from rotation and ellipticity.

By the switch HETEROG the effects of lateral heterogeneity are taken into account.

Table 9.4:

DEFINE (V1.1)	Define the calculation parameters
ED_TIDEBANDS	Edit desired body tide bands
ALL_TIDES	Use the complete body tide model
CLR_TIDES	Clear the complete body tide model
ED_R_DEGS	Edit harm. degrees of effects of rotation & ellipticity
ED_H_DEGS	Edit harm. degrees of effects of lat. heterogeneities
QUANTITIES	Menu for selecting the desired output quantities
POSITION_TYPE	Select type of positions
ED_POSITIONS	Menu for editing the spatial positions
SH_GROUPS	Show tidal groups of the body tide potential

9.3 SET_TIME: Set epoch interval, time step, and time units

All information on this menu is given in Section 2.3.1.

9.4 DEFINE: Define parameters for the calculations

The definition of specific computational parameters is carried out in the DEFINE menu. The available commands and submenus are displayed in the next table.

The ED_TIDEBANDS submenu serves for the definition of a desired subset of tidal constituents of the specified harmonic tide model. It does not seem reasonable to always use the full extent of a tide model. Therefore, the user is enabled to specify so-called tidal bands. A tidal band is constituted by a number of partial tides which (1) belong to the same harmonic degree and order of the tidal potential and (2) have received subsequent partial tide numbers according to their argument indices. The specific harmonic degree and order have to follow the ED_TIDEBANDS command and are separated by a comma. To edit tidebands of degree 2 and order 1, for instance, the user has to enter

> ED_TIDEBANDS 2,1

The specification of the tide bands is achieved by an edit menu for an one-dimensional array of integers which is more generally described in Sect. 2.3.2. As a band is defined by subsequently entering a lower and upper partial tide number the total number of values to be entered must be twice the number of tide bands! It is necessary to start with the lowest partial tide numbers and to proceed with continuously with increasing numbers. To have an overview of the complete tidal model comprising the partial tide numbers it is very useful to possess an according printout or to view the tide model in the VIEW menu!

Use the ALL_TIDES command to select all partial tides present in the body tide potential.

CLR_TIDES will clear the body tide potential so that not a single partial tide is active. This command is particularly useful before defining a new set of tidebands.

ED_R_DEGS invokes a menu for editing those harmonic degrees related to the perturbation effects

Table 9.5:

QUANTIS (V1.0)	Selecting the quantities to be calculated
P_POTENTIAL	Primary tide generating potential
S_POTENTIAL	Secondary tidal potential
P_ACCELERA	Primary tidal acceleration (in spherical components)
S_ACCELERA	Secondary tidal acceleration (in spherical components)
DISPLACEMENTS	Surface displacements (vertical, N-S, and W-E components)
TILT	Surface tilt (N-S and W-E components)
N_GRAVITY	Surface normal gravity

of rotation and ellipticity which will be included in the calculations. Automatically, the associated harmonic orders are included ranging from -n to n if n is a selected harmonic degree.

ED_H_DEGS invokes a menu for editing those harmonic degress related to the perturbation effects of lateral heterogeneities which will be included in the calculations. As in the ED_R_DEGS menu the associated harmonic orders are included automatically.

QUANTITIES invokes a menu for selecting the desired tidal output quantities by according switches.

POSITION_TYPE provides a menu for selecting the type of position the user wants for his calculations.

ED_POSITIONS invokes a menu to edit positions according to the type of position that has been selected. If surface positions have been selected only the geocentric longitudes and latitudes have to be edited as the radii are uniquely determined. It is important to note that for any pair of positional coordinates first the geocentric longitude and then the latitude has to be entered. Thus, the surface position editing is an editing of a two-dimensional array which more generally is described in Sect. 2.3.2. If positions in the outer space have been selected the user additionally has to specify the geocentric radius. In this case, the order of entering a position is defined to be geocentric latitude, longitude, and radius. The general description is to be found in the above mentioned Section.

SH_GROUPS shows the definitions which have been performed concerning the tidebands.

9.4.1   QUANTITIES: Select quantities to be calculated

QUANTITIES is a menu for selecting the desired tidal output quantities by according switches. An overview of the menu is given in the next table:

P_POTENTIAL is a switch for the primary tide generating potential to be calculated according to the extent of the selected tidal constituents. The gradient of this potential is available by the P_ACCELERA switch. It is given by the spherical components. S_POTENTIAL is a switch for the secondary tidal potential with S_ACCELERA being the associated acceleration given in spherical coordinates. DISPLACEMENTS is a switch for the surface displacements given by its vertical, north-south, and west-east components, respectively. TILT gives a switch for the surface tilt which has two components, north-south and west-east. N_GRAVITY is a switch for the surface normal gravity.

Table 9.6:

POS_TYP (V1.0)	Selecting the type of position
SURFACE	Surface positions at the Earth model
OUT_SPACE	Positions in the outer space

Table 9.7:

VIEW (V1.0)	View inputs and results
SCREEN	Set screen output to [Y/N]
LIST_FILE	Set list file output to [Y/N]
RESULTS	Show the available results
TID_POT	Show complete body tide potential model
LOVE_STATIC	Show complete static body tide Love-Shida numbers
LOVE_DYNAMIC	Show dynamic input body tide Love-Shida numbers

9.4.2   POSITION_TYPE: Select the type of positions for the calculation

POSITION_TYPE is a menu for selecting the type of position the user wants for his calculations. The menu is shown in the next table.

The user may only select one of three available position types. The SURFACE switch is used to select positions at the surface of the structural model. By OUT_SPACE positions in the space outside the structural model are selected.

9.5   The VIEW Menu of GEOTIDES

General information on the VIEW menu is available in Sect. **??**. The commands of the VIEW menu are displayed in the next table.

RESULTS will show the results of the calculations.

The remaining commands deal with the inputs of GEOTIDES. TID_POT starts the viewing of the active tide generating potential. This is very useful for a subsequent editing of the tide bands in menu DEFINE.

LOVE_STATIC displays the active static body tide Love-Shida numbers, LOVE_DYNAMIC the dynamic ones.

9.6   The WRITE Menu of GEOTIDES

General information on WRITE is available in Sect. 3.4. The commands of WRITE are given in the next table.

With RESULTS, all results in the time interval specified by T_LOW and T_UP just in the same way as in the VIEW menu (see previous section) are written to the output file.

Table 9.8:

WRITE (V1.0)	Writing the output files
RESULTS	Write results from time interval to file
T_LOW	Set lower bound of time interval in DT (y,m,d,h,min,s)
T_UP	Set upper bound of time interval in DT (y,m,d,h,min,s)
INFO	Display further information

9.7 The WR_GNUPLOT Menu of GEOTIDES

Documentation on this menu is not yet available.

Chapter 10

CONVOLUTE: Global Deformations by Loading

CONVOLUTE is a programme for the computation of quantities related to global deformations which are induced by global loading at the Earth's surface like atmospheric pressure and ice sheets. Though not explicitly required, the input load history should be adjusted such that the spatial load distribution is approximately equidistant in the sense of a length unit. The output quantities are computed from two basic inputs: a set of Green's functions (which were derived from Load Love-Shida numbers) and a load history. The Green's functions may be either time-independent (i.e. purely elastic) or time-dependent (i.e. viscoelastic). The load history must be given in terms of a fixed set of load positions and a fixed number of time values.

CONVOLUTE computes the loading response of an spherically-symmetric Earth model for any position of observation by a convolution of a Green's function with the load history. The algorithm for the convolution works as follows: Starting from the defined position of observation which serves as 'North Pole' in the oberserver's spherical coordinate system, a mesh of meridians and colatitude circles is drawn after the user's specification. This mesh divides the spherical surface into a set of spherical rectangles which are parametrized by polar angle and azimuth counted from the observer's position. Irrespective of time, the Green's functions only depend upon the polar angle. Inside the active spherical rectangle the Green's function is approximated by a polynomial of polar angle, the load function is approximated by a polynomial of polar angle and azimuth. Both sets of polynomial coefficients are determined by a least squares algorithm. Once having these coefficients, the convolution of Green's function and load function inside the rectangle is straightforward. In this manner the complete surface of the Earth is scanned and polynomial fits for Green's functions and load are performed rectangle by rectangle. The final result is obtained by summing up the individual contributions from each of the single rectangles.

The table below shows the main menu of CONVOLUTE:

The menu ED_POS_OBS is used to enter the desired positions of observation. ED_POS_OBS is a standard edit menu which is explained elsewhere. A position is entered as a pair of geographical longitude and latitude in degrees. The remaining menus are described in the following sections.

Table 10.1:

Menu CONVOLUTE (V1.0)	Main menu of CONVOLUTE
FILES	Open files for I/O
SHOW	Show overview of current values of parameters
DEFINE	Define parameters for convolution
ED_POS_OBS	Edit positions of observation for convolution
SELECT	Select results to be calculated
CALCULATE	Perform the convolution
VIEW	View and/or print results and other parameters
WRITE	Write results to output file
INFO	Display further information about menu

Table 10.2:

Menu FILES (V1.0)	Open files for I/O
LOAD_HISTORY	Open input file for model of load history
GREENS_FCTS	Open input file for Green"s functions
OUTPUT	Open output file for the results

10.1   The FILES Menu of CONVOLUTE

The FILES-table shows the three commands to open files for I/O. The file name is entered after the command names. By LOAD_HISTORY the input file containing the load history will be opened for reading. The Green's functions file is activated by the GREENS_FCTS command. To open a file for output of results, use OUTPUT.

10.2   The SELECT Menu of CONVOLUTE

This menu serves for selecting the desired quantities to be computed. The information given in the following table should be self-explanatory.

10.3   The DEFINE Menu of CONVOLUTE

The parameters to be used in the course of CONVOLUTE's CALCULATE execution are assigned in the DEFINE menu which is shown in the table below. The commands START_P_ANGLE and P_ANGLE_STEP deserve a special discussion as they strongly refer to the way how the spherical rectangular surface elements are scanned in the calculations. A rough discussion of the principle algorithm has been given above. To understand the use of the START_P_ANGLE command one has to know that the surface integration starts at an polar angle which must be greater than 0. This start value marks a small circle about the present position of observation: all surface elements outside are scanned in the way already described, the disk-shaped area limited by the small circle however is

Table 10.3:

Menu SELECT (V1.0)	Select quantities to be calculated
VERTICAL	Vertical displacement
TANGENTIAL	Tangential displacement
E_RR	Radial strain [eps(r,r)]
E_TT	Horizontal strain [eps(theta,theta)]
E_LL	Horizontal strain [eps(lambda,lambda)]
E_RT	Strain [eps(r,theta)]
TILT	Deformational part of tilt
GRAVITY	Deformational part of gravity
FREE_AIR	Free air gravity anomaly
GEOID	Geoidal height deformations
SEA_LEVEL	Sea level changes
SHOW	Show current select status
INFO	Display information about items

treated in a special way. All load values falling inside this disk will be averaged and handled as a uniform disk load, its contribution will be evaluated analytically. Note, that the start value of polar angle has to be entered as log10 of angle in degrees.

By P_ANGLE_STEP the integration step concerning the polar angle variable is defined as log10 of angle. The actual step with for polar angle integration thus varies exponentially. Only the azimuth step will be constant, it is defined by the AZIMUTH_STEP command with the angle given in degrees.

The maximum polynomial degrees for fitting the Green's functions and the load are defined by the commands GREEN_DEGREE and LOAD_DEGREE.

The time interval for which the output quantities will be computed is entered by the TIME_INTERVAL command while the time step is defined by the T_STEP command. If the specified time interval and time step do not coincide with those of the load history or if a finer temporal sampling has to be achieved, this values will be linearly interpolated from the time values of the load history. To activate this option, the INTERPOLATE_T switch has to set to Y. The default of INTERPOLATE_T is N. The last of the define commands, RADIUS, presently has no meaning.

10.4   The VIEW Menu of CONVOLUTE

This menu enables to view the input Green's functions, the load history, and the results of the calculations. SHOW gives an overview of parameters for the proper use of the commands of this menu, like the number of temporal samples of the Green's functions and load history, the number of related spatial samples and the code numbers for the appropriate Greens's functions numbers, e.g. 1 stands for the Green's functions dealing with vertical displacements. LIST_FILE and SCREEN serve for defining where the output of this menu should go, to the list file or the screen or both.

A group of five commands is used to view information related to the Green's functions: T_GREEN, PGREEN_P_ANGL, PGREEN_TIME, GREEN_P_ANGL, and GREEN_TIME. T_LOAD will show

Table 10.4:

Menu DEFINE (V1.0)	Define parameters for calculation
INFO	Display information about menu
SHOW	Show current definitions
START_P_ANGLE	Enter start value of polar angle for integr. (log10 of deg)',
P_ANGLE_STEP	Enter step of polar angle for integration (log10)
AZIMUTH_STEP	Enter step of azimuth in degrees for integration
GREEN_DEGREE	Enter polynomial degree of approx. for Green''s function
LOAD_DEGREE	Enter polynomial degree of approx. for load function
TIME_INTERVAL	Enter time interval for integration
T_STEP	Enter time step for integration in hours
INTERPOLATE_T	Time interpolation of load history and Green's fcts [N/Y]
RADIUS	Enter radius in km for which the functions are calulated

the times for which the input Green's functions are given. PGREEN_P_ANGL followed by a specific Green's functions and time number shows the polar-angle dependent polar Green's function for the specified time. These Green's functions are used for the computation of the effect of the disk load at the position of observation. PGREEN_TIME followed by a specific Green's function and polar angle number shows the time-dependent polar Green's functions. GREEN_P_ANGL and GREEN_TIME work in exactly the same way, however, the normal Green's functions will be shown.

Another group of four commands is related to view the load history. These commands are: POS_LOAD, T_LOAD, LOAD_POS, and LOAD_TIME. POS_LOAD will give an overview of all load positions, T_LOAD of all times of the load history. LOAD_POS followed by a time number lists the position-dependent values of the load history. To view the time-dependent load history use the LOAD_TIME command followed by a position number.

Finally, the RESULTS command (followed by a number for the specific quantity) displays the results obtained by CONVOLUTE's calculation.

An overview of the available commands is given in the table below:

Table 10.5:

Menu VIEW (V1.0)	View load, Green's numbers, and results
SHOW	Show parameters for the use of the below commands
LIST_FILE	Set list file output to [Y/N]
SCREEN	Set screen output to [Y/N]
T_GREEN	Show times for which the Green's functions are given
PGREEN_P_ANGL	Show polar angle-dependent polar Green's function
PGREEN_TIME	Show time-dependent polar Green's function
GREEN_P_ANGL	Show polar angle-dependent Green's function
GREEN_TIME	Show time-dependent Green's function
POS_LOAD	Show geographical positions of load
T_LOAD	Show times for which the load function is given
LOAD_POS	Show position-dependent load function
LOAD_TIME	Show time-dependent load function
RESULTS	Show the results

Chapter 11

SEALEVEL: Solution of the Sea-Level equation

Chapter 12

GYROS: Integration of the equations of rotation

GYROS estimates Earth rotation parameters (ERP) as function of time by integration of the equations of terrestrial rotation (ETR). The ETR allow for a number of possible exogenic excitations. The structural model of the rotating planet includes a fluid outer core and (not yet!) a solid inner core.

The main menu of GYROS is given in the next table.

12.1   FILES: Open input and output data files for GYROS

The structure of FILES in GYROS is given in the next table.

Table 12.1:

GYROS (V1.1)	Integration of the equations of rotation (ETR)
FILES	open files for I/O
SHOW	Short overview of current parameters
EXCITATIONS	Design, determine, or read exogenic excitation functions
DEFINE	Define parameters for the integration
SELECT	Select the extend of the ETR and the excitations
CALCULATE	Determination of ERP
VIEW	View and/or print results and other parameters
WRITE	Write results to output file

Table 12.2:

FILES (V1.1)	Opening I/O files for GYROS
OBSERVED	Open input file for observations of ERP
ATMOSPHERE	Open meteorological data file for input
OCEAN	Open description file for ocean model
STRUCTURE	Open description file for the structural model
LOVE_GREEN	Open input file with Love Numbers and Green's functions
ALT_EXCIT	Open input file which contains alternative excitations
EXCITATIONS	Open output file for time series of excitations
ERP	Open output file for ERP series
COMPLETE	Open output file for a complete set of series

Table 12.3:

EXCITAT	Exogenic Excitation of rotation
SHOW	Show current status
DEFINE	Define parameters for data preparation and excitation functions
CREATE	Create synthetical excitation data
INERTIA	Calculate time series of atmospheric moment of inertia
DESIGN	Select the design of the loading functions
LOADING	Calculate the loading function
SPHERICALS	Extract time series of the required spherical harmonic terms
SELECT	Select excitation functions to be calculated
COMPOSE	Create series of excitation functions
LOAD	Load any of the time series
VIEW	View results
WRITE	Write any of the time series

12.2   EXCITATION: Design, determine or read exogenic excitations

12.2.1   DEFINE: Define data preparation and excitation functions

12.2.2   CREATE: Create synthetical data for excitation functions

12.2.3   DESIGN: Design the load functions

12.2.4   SELECT: Select the exogenic excitations

12.2.5   LOAD: Load time series of exogenic excitations

12.2.6   VIEW: View exogenic excitations

12.2.7   WRITE: Write exogenic excitations

Part IV

POTS: Processing Of Time Series

Chapter 13

FILREC: Design of transversal filter operators

Chapter 14

FILTER: Filtering or smoothing of time series

Chapter 15

MOVEMEAN: Data reduction by moving averages

Chapter 16

CORREL: Correlation of time series and discret functions

Chapter 17

FOURIER: Calculation of Fourier spectra

Chapter 18

VARSPEC: Variance spectra and variance analyses

VARSPEC is a utility serving for the computation of variance, amplitude and phase spectra and performing variance analyses using preliminary results. Fitting harmonic models to data is also possible as well as searching for harmonic constituents given the number of signals to be looked for, start values for each and the frquency range.

A discussion of the properties of the variance spectrum can be found in Plag (1988).

For these purposes a set of commands and submenus is available as is listed in the table below.

For the calculation of variance spectra different function types can be chosen in submenu INIMOD (see Sect. 18.4.1), among them damped harmonis with random excitation. The most common case, that of undamped harmonic oscillations, is described in detail below.

VARSPEC applies an extended version of the simple variance spectrum proposed by Vanicek (1970), which was developed by Plag (1988).

Table 18.1:

VARSPEC	Variance spectra, variance analyses and fits
SHOW	Show current status
SET_TIME	Set time interval, units, and origin
FILES	Open input and output files
EXTRACT	Read data into data vector
TRANSFORM	Transform data into desired form
SPECTRUM	Define and calculate next spectrum
ANALYSE	Analyse current spectrum
MODEL	Edit extend and parameters of current model
ADJUST	Adjust non-linear parameters of model
RESIDUAL	Calculate residual time series using current model
VIEW	View selected results, including model and spectrum
WRITE	Write selected results to output
INFO	Display additional information about menu

Let $X = x(t_i)$, $i = 1, \ldots, N$ be a not necessarily equidistant time series, with the variance $v(X)$ being defined in the usual way, i.e.

$$v(X) = \frac{1}{N-1} \sum_{i=1}^{N} (x(t_i) - \bar{X})^2, \tag{18.1}$$

where $\bar{X}$ is the arithmetic mean value of all $x(t_i)$, $i = 1, \ldots, N$. For a frequency-dependent base function $Y_\omega = y_\omega(t, \vec{p}(\omega))$ which is linear in $\vec{p}$, we may, for any value of ω, calculate the vector $\vec{p}(\omega)$ using a least squares fit of Y to X. With the residual $R_\omega = X - Y_\omega$, we may define the variance spectrum $V\{X\}$ as

$$V\{X\}(\omega) = \left(1 - \frac{v(R_\omega)}{v(X)}\right). \tag{18.2}$$

Usually, $V\{X\}$ is given in per cent. The simple variance spectrum used by Vanicek (1970) utilizes the base function

$$y_\omega(t, \vec{p}(\omega)) = a\sin(\omega t) + b\cos(\omega t) + c, \qquad \vec{p}(\omega) = (a(\omega), b(\omega), c(\omega)), \tag{18.3}$$

i.e. a simple harmonic oscillation, and $V\{X\}$ may be estimated for any value of ω within the interval $[\frac{1}{2(t_N - t_1)}, \frac{1}{\overline{\delta t}}]$, where $\overline{\delta t}$ is the mean sampling interval. Plag (1988), in his extended version, used a more general base function, namely

$$y_\omega(t, \vec{p}(\omega)) = \sum_{j=1}^{M} (a_j \sin(\omega_j t) + b_j \cos(\omega_j t)) + \sum_{l=0}^{P} c_l t^l + a\sin(\omega t) + b\cos(\omega t), \tag{18.4}$$

where the first sum in (18.4) accounts for M harmonic constituents of fixed frequencies ω_j, the second sum models any drift in X with a polynomial of degree P and the remaining term accounts for a harmonic variation of variable frequency ω. The vector $\vec{p}$ depends, of course, on the free frequency ω, and it now has the form

$$\vec{p}(\omega) = (a_1(\omega), b_1(\omega), \ldots, a_M(\omega), b_M(\omega), c_0(\omega), \ldots, c_P(\omega), a(\omega), b(\omega)), \tag{18.5}$$

and $\vec{p}(\omega)$ ist estimated as before.

The variance spectrum defined in (18.2) has the advantage to be a time–domain method, which may be used in the presence of gaps without requiring an interpolation of these gaps, and even for non-equidistant data the variance spectrum may be computed. Furthermore, the resulting spectrum is not significantly affected by small gaps.

By using the principle of superposition the method reveals additionally an amplitude spectrum as well as a phase spectrum.

With the submenu EDIMOD (see Sect. 18.4), which is called using the command MODEL, the M known constituents and the maximum degree P of the trend functions included in the base function 18.4 can be defined.

Variance analyses may be perfomed taking the following path:

1. First a primary variance spectrum is claculated.

Table 18.2:

FILES	Open specific files
INPUT	Open input data file
BASE	Open input file for data base (for ANALYSE)
SPECTRUM	Open output file for variance spectrum
PEAKS	Open output file for peak data
MODEL	Open output file for current model
ALL	Open output file for original-, model-, residual time series

2. This then is searched for all peaks exceeding a distinct level which the user has to define. The respective frequencies, amplitudes and phasese are used to set up a new base function. Submenu INIMOD offers the utilities needed for this task (see Sect. 18.4.1).

3. Now a secondary variance spectra is calculated, using the base function determined in step 2.

4. To refine the results, it may be appropiate to repeat steps 2. and 3. several times.

5. Finally the fixed frquencies are further adjusted in an iterative process, and the parameters of the base function are determined in a simultaneous fit. This is done by using the FIT command which is part of ADJUST (see Sect. 18.5).

If a model was defined and adjusted a residual time series can be computed by calling RESIDUAL. This call results in substracting the current model from the original data.

The command SHOW and the menus called by, SET_TIME, EXTRACT and TRANSFORM are standard facilities and therefore described in the Chapters 2 and 3. The remaining commands are calls to submenus described in the following sections. It should be mentioned that the commands not necessarily compare to the respective menu names. The two which do not are ANALYSE (calls VARANA) and MODEL (calls EDIMOD).

18.1 FILES: Open specific files

The general structure and purpose of FILES is described in Sect. **??**. However, there are some specific commands as listed in the table below, which will be explained in this section.

BASE is not implemented yet. SPECTRUM opens a file determined for the calculated spectra. Its content depends on the number M of known constituents which are to be chosen in submenu MODEL. In detail the file's columns will contain: $F, V, V_{C_1}, \ldots, V_{C_M}, A, A_{C_1}, \ldots, A_{C_M}, P, P_{C_1}, \ldots, P_{C_M}$, where F denotes for the frequency, V for the variance, A for the Amplitude, P for the phase and the indizes for the differnt model constituents.

If the user wants to analyse the computed spectrum PEAKS offers the possibility to open a file to which the results can be stored. One record of such a file will contain

1. (...),

Table 18.3:

SPECTRM	Calculating spectra
SHOW	Show current status
DEFINE	Define parameters of spectrum
CALCULATE	Calculate spectrum
INFO	Display additional information about menu

2. the peaks frequencies,

3. its variance,

4. (...),

5. (...),

6. the rspective amplitude value,

7. (...),

8. the respective value in the phase spectrum,

9. (...),

10. (...) and

11. (...) .

MODEL

ALL is intended to open a file of which each record contains the time, the original value, the model value and the residual.

18.2   SPECTRM: Calculating spectra

SPECTRM serves for calculating a variance spectrum. Actually this task is performed by the command CALCULATE. But before calling it some parameters like frequency step and range have to be defined using the submenu DEFSPE. It is called by DEFINE and its structure is discussed in a seperate subsection (see Sect. 18.2.1).

18.2.1   DEFSPE: Defining some parameters necessary for calculating spectra

The commands of DEFSPE as listed in the table are designed to define the parameters of a spectrum before calculating it.

UNITS is not installed. Instead of this the frequency units are determined automatically from the time units. AUTOMATIC asks whether or not the parameters (frequency step and interval etc.) should be calculated by the program.

DEFSPE	Definig parameters of spectrum
UNITS	Enter frequency units
AUTOMATIC	Select automatic determination of spec. params.
STEP	Frequency step for spectrum
INTERVAL	Frequency interval for spectrum
KIND	Kind of spectrum
Q	Enter Q for variable frequency, if damped
F_MIN	Enter minimum distances to known frequencies
COLUMNS	Edit indexes of columns to be used for spectrum
MODEL	Use model (if any defined) [Y/N]
INFO	Display additional information about menu

If this is denied STEP and INTERVAL are used to fix the step width and the bounds of the interval respectively. Doing this the following should be considered:

Let L be the length of the time series and $\overline{\delta t}$ the average sample interval. Both are given in time units. Then the lower and upper bound, f_l and f_u (in cycles per time unit), of the frquency band which can be analysed are determined by

$$f_l = \frac{1}{L} \quad \text{und} \quad f_u = \frac{1}{\overline{\delta t}}.$$

Two harmonic constituents with the frquencies f_1 and f_2 can be resolved provided that

$$|f_1 - f_2| > \frac{1}{2L}.$$

To get a optimum spetrum the step should always be less than $\frac{1}{2L}$, let's say not more than a fifth of it.

KIND is a submenu which provides the possibility to choose the kind of the function. It will be described below.

If the user wishes do calculate a spectrum of **damped** harmonic oscillations the quality factor may be set using the command Q.

In the vicinity of known constituents which were set in the menu EDIMOD (see Sect. 18.4) the calculations are unstable. Thus a minimum distance to these frequencies should be set using F_MIN.

With COLUMNS the components of the work file are selected which are to be used for the calculation of the spectrum. Actually COLUMNS is a call to a standard edit menu as it was described in Chapter 2.

Finally calling MODEL the user has to tell the program whether or not he wants to include a model to the base function. The model should have been defined before otherwise a error message would occure.

SPEKIND: Determining the kind of the base function

The menu SPEKIND serves for selecting the function's type out of a set of possibilities which is listet in the table.

Table 18.5:

SPEKIND	Determining the kind of the base function
HARMONIC	Normal variance spectrum, using sine and cosine
CIRCLE	Variance spectrum of harmonic rotation
ELIPSE	Variance spectrum of eliptic rotation
DAMPED	Variance spectrum of damped harmonic oscillation

Table 18.6:

VARANA	Analysing the current variance spectrum
SHOW	Show current status
SPIKES	Find all spikes in spectrum
PEAKS	Calculate peak parameters (area, width)
DEFINE	Define model peak
CORRELATE	Correlate model peak with spectrum
INFO	Display further information about menu

18.3   VARANA: Analysing the variance spectrum

If a spectrum was calculated the menu VARANA enables the user to analyse the results by using the commands listed in the table. However, DEFINE and CORREALTE are until now not implemented.

First the peaks have to be searched. For this serves SPIKES. Afterwards their parameters like width and area can be calculated using PEAKS. The results may be written to a file (see Sects. 18.1 and 18.7).

18.4   EDIMOD: Defining a model

The menu EDIMOD is designed to define a model which lateron can be included to the base function. For this purpose a number of commands an submenus is available as listed in the table below. The submenus for editing the parameters are of the standard types descriped in Sect. 2.2. Therefore their structure is not discussed in this section. But it should be mentioned that the units of the parameters must fit to the units of the respective ccolumns of the work file, i.e. if the time is given in years the frequency unit must be cycles per year etc.

First the number of constituents has to be set by using NUMBER. The following submenus serve for editing their different parameters.

With DAMPED the user can choose whether some of the constituents should be a damped harmonic oscillation and Q serves for defining the respective quality factors. The defaults are NO and -1.0 respectively.

If the model contains damped harmonics the individual times of first excitation have to be set using the FIRST submenu. The program provides a stochastic excitation mechanism, i.e. the oscillations

Table 18.7:

EDIMOD	Defining a model
SHOW	Show current status
NUMBER	Set number of known signals
PERIOD	Edit periods of signals
FREQUENCY	Edit frequencies of signals
AMPLITUDE	Edit amplitudes of signals
PHASE	Edit phases of signals (in degrees)
DAMPED	Edit kinds of signals (harmonic or damped)
Q	Edit quality factors for damping (E**(-W*T/TAU(Q))
FIRST	Edit time for first excitation for damped harmonics
EXITATION	Edit excitation level for damped harmonics
TREND	Enter degree of polynomial used trend function
COEFFICIENTS	Edit coefficients for trend
INITIALIZE	Initialize model using peak data
INFO	Display additional information about menu

Table 18.8:

INIMOD	Initializing a model using peak data
SHOW	Show current status
LEVEL	Set level of variance for peaks to be used
SEARCH	Search all peaks above level
INFO	Display additional information about menu

are excited each time a random variable exceeds a given level. This variabel varies between 0.0 and 1.0 and the levels are set using EXCITATION.

Using TREND the degree of the trend polynomial is set and COEFFICIENTS allows to define its coefficients.

INITIALIZE is a call to the submenu INIMOD, which is described in detail in a seperate subsection. It serves for constructing a model using primary results in the case variance analyses are to be performed.

18.4.1   INIMOD: Initializing a model using peak data

If the user wants to carry out variance analyses first a primary spectrum has to be calculated as was mentioned above. Then the menu INIMOD supplies the possibility to analyse the spectrum and to initialise a new model.

For that purpose the user has to set a variance level by the command LEVEL first. A call of SEARCH then results in selecting all peaks of the current spectrum whose variance exceeds this level and uses them to initialise the model by taking the respective values of the amplitude and of the phase spectrum.

Table 18.9:

ADJUST	
SHOW	Show current status
RESET	Reset model and run parameters
LOAD	Load old model
DEFINE	Define parameters for adjusting
FIT	Fit model to data
UPDATE	Update model parameters
INFO	Display additional information about menu

18.5   ADJUST:

ADJUST offers two possibilities of fitting a model to the data. The first is that the frequencies defined in EDIMOD remains fix while phase and amplitude are optimised by a least square fit. This task is performed by UPDATE.

The second method is a more sophisticated one resulting additionally in optimised frequencies. In an iterative process each frequency is changed stepwise, and the parameters are determined anew. For each frequency this is repeated until the variance is at its maximum. When all frequencies are adjusted, the complete procedure is repeated again, until no further corrections are found for any model constituent. This method is usually the last step of a variance analysis. The described process is started by calling FIT.

But before starting the fit some parameters should be defined. For this purpose the menu DEFADJ (see Subsect. 18.5.1) is called by DEFINE.

If the user wants to reset the current model to start again with a once defined model, he or she may call RESET. And if he or she wants to use an old model which was saved to the back-up file, LOAD may be used.

18.5.1   DEFADJ: Defining parameters for adjusting

DEFADJ serves for defining some parameters needed for the fits performed by ADJUST.

Q??

The stepwidth with that the frequencies are varied is set by using STEP while PRECSION allows to determine the tolerance of the iterative process. LOWER_LIMIT sets the minimum amount of sample points required to perform the fit.

18.6   VIEW: Displaying selected results

The general structure of this menu is described in Subsect. **??**. However, there are some non-common commands which are listed in the table below. The comments listed there are sufficient to describe them.

Table 18.10:

DEFADJ	Defining parameters for adjusting
Q	Enter factor for adjusting Q
STEP	Enter step for adjusting frequency
PRECISION	Enter relative precision level
LOWER_LIMIT	Enter lower limit of available data required for adjusting
INFO	Display additional information about menu

Table 18.11:

VIEW	Displaying selected results
SPECTRUM	Display spectrum
SPIKES	Display list of spikes in spectrum
PEAKS	Display list of peak parameters for spikes
MODEL	Display parameters of current model
DATA	Display data including model data and residual
LIST_FILE	Set list file output to [Y/N]
SCREEN	Set screen output to [Y/N]
INFO	Display information about menu

18.7   WRITE: Writing slected results to output files

The submenu WRITE is designed to store results. Commands listed in the below table refer to respective files opened by using the FILES menu. Theire content is described in Sect. 18.1.

Table 18.12:

WRITE	Writing selected results to output files
SPECTRUM	Write file containing the spectrum
PEAKS	Write file containing the peaks
MODEL	Write file containing model parameters
ALL	Write file containing data, model and residual
INFO	Display information about menu

Chapter 19

FITALL: Fit of linear and non-linear models to data

Chapter 20

TRANTIM: Calculation of transfer functions in the time domain

The programme TRANTIM is used to calculate transfer functions in the time domain. The transfer functions are determined by fitting simultaneously a number of 10 model input (MI) parameter to one additional parameter, named the Real System Output (RSO). The fit results in one time dependent transfer function for each of the MIs and the model output which can be compared to the RSO.

The menue of TRANTIM is shown in the next table:

Before the calculations may be performed, the model configuration has to be stored into a work file. This is a difficult task which may cost a lot of CPU time. Therefore, the correct procedure of creating the required work file is now described in detail.

At first some parameters of the new model have to be defined. This requires three initial steps to be performed: The RSO data file has to be opened by calling to FILES. The time interval, the time origin, and the time units of the data for which the transfer functions will be calculated have to be set by calling to SET_TIME. The number of MIs has to be defined by calling to DEFINE. DEFINE is a

Table 20.1:

TRANTIM (V1.0)	The TRANTIM main menu
SHOW	Show short overview of current parameters
FILES	Open files for I/O
SET_TIME	Set time interval, origin or units
CREATE_WRKF	Create work file for data matrix
EXTRACT	Extract data from input file
TRANSFORM	Transform data in work file
SAVE	Save work file
DEFINE	Define sample points of transfer function
CALCULATE	Calculate transfer function(s)
VIEW	View and/or print contents of work file
WRITE	Write results to output file
INFO	Display additional information about menu

Table 20.2:

Column number	Column entry
1	time
2	RSO
3	MI 1
.	.
.	.
m + 2	MI m
m + 3	model output
m + 4	residual

standard menu for editing integer arrays, and the dimension of the arrays set in DEFINE equals the number of MIs.

Now, the programme is ready to create the structure of the work file. This is done by entering the command CREATE_WRKF. The structure of a work file for m MIs is given in the next table:

The time axis is generated during the creation of the work file. It is essential that the data files are read into the columns 2, 3,..., m+2 according to the given scheme! The reading itself is done by opening the respective input data file in the FILES menu and extracting its data by calling to EXTRACT. The columns m+3 (model output) and m+4 (residual) will be filled during the calculation of the transfer functions.

In the current version of TRANTIM the number of MIs is limited to 10. If all these input channels are used, up to 4545 data points may be read in for each of the input channels. If the number of MIs is smaller, the maximum number of data points increases to 5000. These limitations may easily be raised by changing the respective parameters in the FORTRAN code.

As the creation of the work file may take some time, it seems reasonable to save it as a regular data file, although it is kept after leaving the programme. The saving can be done by creating an output data file (menu FILES) and entering the command SAVE. As the created data file contains the model configuration in the correct way, during a later call to TRANTIM it can be read in on a whole by using EXTRACT without CREATE_WRKF.

When all the required data are stored into the work file, the calculation of the transfer functions may be started. Therefore, in the DEFINE menu the user may specify the shift parameters used for the calculation. For each of the MIs three parameters have to be entered, namely the lower shift, the upper shift, and the step width for the shifts. A zero step width is not allowed, but the programme checks for this case.

Now the transfer functions are calculated by calling to CALCULATE.

The results are kept in the work file or in arrays, and they may be viewed or written to output files using VIEW or WRITE.

20.1   FILES: Open data files for input and output

The structure of the FILES menu is shown in the next table.

Table 20.3:

FILES V(1.0)	Opening files for TRANTIM
RSO	Open input file for Real System Output
INPUT	Open input file for model input
TRANSFER	Open output file for transfer functions
RESULTS	Open output file for model results
WORK_FILE	Open output file to save work file

Table 20.4:

VIEW (V1.0)	View data matrix and results
LIST_FILE	Set list file output to [Y/N]
SCREEN	Set screen output to [Y/N]
WORK_FILE	Display content of work file
TRANSFER_F.	Display transfer functions
INFO	Display information about menu

Although the RSO and the MI data files do not differ in principal, they are opened separately by the two commands RSO and INPUT. This is necessaray since the description parameters of the RSO data file are handled individually within the programme.

TRANSFER opens the output file for the transfer functions. All transfer functions of the model configuration are stored into one file. If the model consists of m MIs, the TRANSFER data file takes 2m columns. For each of the MIs there is one column containing the shifts and one containing the respective transfer function coefficients.

RESULTS opens the file for the model output. It carries four columns: time, RSO, model output, and residual.

How to use WORK_FILE was explained within the main menu.

20.2 VIEW: Display the data and the results

The commands available within VIEW are shown in the next table.

Using the VIEW menu the content of the work file and the calculated transfer functions can be displayed on the screen or written to the list file. When a calculation is performed the model output and the residual are stored into the work file. So, they are also available within VIEW by displaying the whole work file.

20.3 WRITE: Write results to output files

Two operational commands are available within WRITE. These are shown in the next table.

WRITE (V1.0)	Write results to files
TRANSFER_F.	Write transfer function to output file
MODEL_OUTPUT	Write real system, model and residual fo file
INFO	Display information about menu

By TRANSFER_F. all the transfer function coefficients are written to the disk. The file contains 2m columns if there are m MIs. For each of the MIs the sampling points of the transfer function are written into one column and the respective transfer function coefficients are written into the neighbouring.

MODEL_OUTPUT creates a data file with four columns. The first column contains the time axis, the next three columns contain the RSO, the model output, and the residual.

Chapter 21

SYNTHET: Create synthetical data

Chapter 22

SYNLOAD: Creation of Synthetical Load Histories

SYNLOAD is a programme for the creation of synthetical, temporally variable loads at a planet's surface, either on the global or on any smaller scale; such a space-time-dependent time series is called a load history. The benefit of SYNLOAD is obvious: it makes available user-defined load histories for continuing case studies of loading-induced deformation processes of the planet. Presently, the prime candidate for such studies is the programme CONVOLUTE explained elsewhere in this manual package. SYNLOAD will compute load histories which — in the sense of a direction — exhibit normal forcings of the planet's surface, an extension to a complete forcing in terms of a stress vector or so-called traction is easily achieved by a suitable supplementation of SYNLOAD's source code, however, it is not yet available. SYNLOAD creates a set of load positions which are nearly uniformly distributed over the load covered area in the sense that the distances between any two neighbouring positions are nearly the same. This kind of load distribution has been chosen to avoid positional clustering towards the north and south poles which occurs if the positions were distributed with constant longitudinal and latitudinal spacing.

The features of the SYNLOAD's main menu are shown in the following table:

The main menu offers three menus for selecting general qualities of the load history: SEL_COVER, SEL_SPACE, and SEL_TIME. The general features of the load function are selected in these menus.

Table 22.1:

Menu SYNLOAD (V1.0)	Main menu of SYNLOAD
INFO	Display further information about menu
SHOW	Show overview of current values of parameters
FILES	Open files for I/O
SEL_COVER	Select coverage of load
SEL_SPACE	Select space-dependent function for load history
SEL_TIME	Select time-dependent function for load history
DEFINE	Define values for calculation
CALCULATE	Calculate load history
WRITE	Write results to output file

Table 22.2:

Menu FILES (V1.0)	Open files for I/O
LOAD_HISTORY	Open output file for load history

Table 22.3:

Menu SELCOV (V1.0)	Select coverage of surface
GLOBAL	Use global coverage by load
RECTANGULAR	Use spherically rectangular coverage by load

SEL_COVER concerns the type of coverage of the planet's surface by the load, SEL_SPACE and SEL_TIME refer to the load function as a product of a space and a time-dependent factor which are selected in these menus.

22.1 The FILES Menu of SYNLOAD

There is only one command in the FILES menu: LOAD_HISTORY. It opens an output file which will contain the computed load history.

22.2 The SEL_COVER menu of SYNLOAD

In the menu the extent of the coverage of the planet's surface by the load history is selected, as shown in the next table.

The command GLOBAL will lead to a global load history, the complete surface of the planet will be covered covered by loads. In its present version, SYNLOAD will deny to select a non-global load.

22.3 The SEL_SPACE menu of SYNLOAD

The content of the SEL_SPACE menu is shown in the next table.

This menu allows the selection of exactly one of its entries. With LEGENDRE a space-dependency of the spherical harmonic type is selected. The space functions are the associated Legendre polynomials

Table 22.4:

Menu SELSPC (V1.0)	Select the space-dependent part of the load function
LEGENDRE	Use associated Legendre polynomials
RANDOM	Use spatially and temp. random values for load
USER	User defined function

Table 22.5:

Menu SELTIM (V1.0)	Select the time-dependent part of the load function
CONSTANT	Use temporally constant load
LINEAR	Use linear temporal decay
EXPONENTIAL	Use exponential temporal decay

with full normalization applied. The load thus will display a space dependency only the polar angle (counted from the north pole), no azimuthal dependency will be present. RANDOM will create random values of the load magnitude within given limits. It should be noted that the RANDOM selection not only refers to the space variable, but even to the time variable; any temporal selection made in the SEL_TIME menu will not be in effect, if RANDOM is selected! The USER command will give rise to a space-dependent function which previously has been defined by the user in the source code of SYNLOAD before the compilaton of the executable code. So, no general statement on USER is possible here.

22.4   The SEL_TIME Menu

This menu serves for the selection of the time-dependent part of the load function. In the following table three available commands are shown:

Obviously, the CONSTANT command will invoke a temporally constant load history. The two remaining command must be supplemented by a time parameter which has to be defined in the DEFINE command. By the LINEAR command, a linear decay of the load function will be generated; for the first time step, the load function will start with the maximal amplitude defined in the DEFINE menu, then if the time parameter (simply the slope in the linear case) is negative the last time step will yield zero load function values. A positive time parameter, however, will result in a temporally increasing load function. The EXPONENTIAL command works in a similar way. The time dependence now is given by the exponential function 'exp(ct)', where c is the time parameter and t the time variable. Again it must be stressed that all these selections do not work if space dependency has been selected as RANDOM.

22.5   The DEFINE menu of SYNLOAD

The parameters which are used in the course of the load computation will be defined in this menu, a command overview is given in the next table.

SYNLOAD needs to know the radius of planet for which the load will be computed, it is given in units of kilometres with the RADIUS command; by default, the Earth's radius of 6371.0 km is used. The resolution of the geographical load distribution is controlled by the RESOLUTION command, here a scale-length in kilometres has to be entered; the smaller the scale-length the greater the number of load positions, the bigger the output file.

Table 22.6:

Menu DEFINE (V1.0)	Define parameters for calculation
INFO	Display information about menu
SHOW	Show current definitions
RADIUS	Radius of loaded planet in km
RESOLUTION	Set scale-length of resolution of load in km
AMPLITUDE	Set maximal amplitude of load in Pa
T_INTERVAL	Set time interval for load history in h
T_STEP	Set time step for load history in hours
SPACE_PAR1	Set first space-dep. function parameter
SPACE_PAR2	Set second space-dep. function parameter
TIME_PAR1	Set time-dep. function parameter

An important parameter of the load history is the amplitude of the load, it is entered by the AMPLITUDE command in units of Pascals; if the a random load will be computed, the effect of a defined amplitude A is the following: the possible range of load amplitudes is -A to +A.

The temporal specification of the load history can be defined by the T_INTERVAL and T_STEP commands; times are given in units of hours. By T_INTERVAL the start and final time of the load history are specified, the T_STEP command defines the temporal step. The following example shows how to arrange a load history starting with 3 h lasting up to 100 h with a time step of 2.5 h:

> T_INTERVAL 3.0,100.0

> T_STEP 2.5

The next two commands refer to the definition of the space-dependent part of the load function: SPACE_PAR1 and SPACE_PAR2. Presently, these commands control the harmonic degree and order of the Legendre polynomials: by SPACE_PAR1 the harmonic degree is entered, by SPACE_PAR2 the harmonic order is entered.

Finally, the TIME_PAR1 command is introduced. TIME_PAR1 will be used to enter the necessary time parameter for the time dependence of the load history. This command only has an effect if a linear or exponential time function has been selected. So, TIME_PAR1 is either the linear time parameter or the exponential one, in both cases is must be entered in units of 1/h.

22.6   The WRITE Menu of SYNLOAD

This menu refers to the writing of an output file as specified in the FILES menu. Generally, two kinds of output file formats may be created and these are defined by the COORDINATES command which may be set to N or Y. By default, N is applied, meaning that the coordinates of the load positions will not be written to the output file. This setting is appropriate for using the output file with the CONVOLUTE programme. With the selection COORDINATES Y the geographical coordinates of the load locations will be included in the output file, this file, however, will not be recognized by CONVOLUTE, but it may serve for other purposes. The actual writing of the output file is started by the OUTPUT command. An overview of the WRITE menu is shown in the following table.

Table 22.7:

Menu WRITE (V1.0)	Write output file
INFO	Display further information about menu
SHOW	Show overview of current values of parameters
COORDINATES	Set output of load coordinates and times [n/y]
OUTPUT	Write output file

Chapter 23

TIVARI: Analyses of ocean tides

Chapter 24

ANATIDE: Analysis of Earth tides

Chapter 25

MORPHE: Access to the topography of a planet

Chapter 26

WMODATA: Access to synoptical data files

The programme WMODATA is designed to calculate climatological mean values from synoptical data that have got WMO-format. Different kinds of parameters for the calculation can be defined according to the demands of the user. In the current version the programme generates standard output data files, containing one type of climatological data for one station. In a later version the appending of the created climatological data to the large existing climatological data base shall be realized.

The main menu is given in the next table.

The DEFINE menu enables the user to set the conditions for calculating the monthly mean values. These include the minimum number of days and the number of registrations per day required to calculate the monthly means.

The calculation is realized by calling CALCULATE. The new climatological data are stored in a work file. Therefore, they are available for any output up to the next call to CALCULATE.

There are two comands to influence the amount of data displayed, namely SELECT and SETTIME. By SELECT the meteorological parameters can be specified which will be displayed. SETTIME allows to set a time-interval to which the output will be restricted. The same options are given in the menus VIEW and WRITE where an output of synoptical or climatological data can be performed.

Table 26.1:

WMODATA (V1.0)	The WMODATA main menu
SHOW	Show short overview of current status
FILES	Open files for I/O
DEFINE	Define parameters for calculation
CALCULATE	Calculate climatological means
SET_TIME	Define time parameters for output
SELECT	Select kind of data for output
VIEW	View and/or print results
WRITE	Write data or results to output files
INFO	Display additional information about menu

Table 26.2:

FILES (V1.0)	Opening file for input and output
DESCRIPTION	Open input file for description of stations
SYNOP_DATA	Open input file for meteorological data of one station
READ_DATA	Read synop data from input data file
METEOROLOGY	Open output file for meteorological data of one station
STATION_CLIMATE	Open output file for climatological data of one station
APPEND_CLIMATE	Open output file to append climatological data

26.1 FILES: Open data files for input and output

Two input and three output files can be opened. The menu structure is shown in the following table:

DESCRIPTION opens the station description data file containing information about the meteorological stations. The station description is given in the standard WMO-format. From this data file parameters like station name, longitude and latitude are read in, which are required to create output data files.

SYNOP_DATA opens the input data file containing the synoptical data of one station. This file must not contain a header. The structure has to be the WMO-format.

If the programme was able to open the new synoptical data file, it reads the data from the files and stores them in a work file. During the reading a first selection takes place. The WMO data files contain up to 31 meteorological parameters. In the current version only eight of them are stored. These parameters are: temperature, windspeed, winddirection, air-pressure, air-pressure-gradient, humidity and sea-surface-temperature. If other meteorological data shall be read in, the programme has to be changed. To ease the necessary changes all data-type dependent statements are contained within the modul 'modsel'. In the main programme only the dimensions of the work files have to be adjusted and the read-format contained in 'readat' has to be changed.

During the reading of the synoptic data informations about the station parameters are displayed. These include possible changes in the height of the stations. A history of the height datum is created, which can be used to calculate height-corrected pressure data, if these are not included in the synoptic data file. Also informations about read errors are shown on the display.

METEOROLOGY is used to open an output file containing the selected parameter of the original synoptic data for one station.

STATION_CLIMATE works analogue to METEOROLOGY but on the data files containing the monthly means

Both kinds of output files can be opened automatically. Then a default name will be set, consisting of the first 12 characters of the station name and an extension indicating the data type contained in this file. The default extensions are shown in the next table.

In a later version APPEND_CLIMATE will open the data base for appending the new climatological data.

Table 26.3:

Parameter	meteorol. data file	climat. data file
Temperature	TMP	CTM
Windspeed	WIS	CWS
Winddirection	WID	CWD
Air pressure	SLP	CSL
Air pressure gradient	PGR	CPG
Humidity	HUM	CHU
Sea surface temp.	SST	CST

Table 26.4:

DEFINE	Define parameters
SHOW	Show short overview of current parameters
LENGTH OF WINDOW	Not in this version
MONTHLY MEANS	Window length = one month
REG/DAY	Edit array of sample points
DAYS/MONTH	Set minimum number of days per month
INFO	Display informations about menu

26.2 DEFINE: Set parameters for calculation

The DEFINE menu enables the user to define the conditions for the calculation of climatological means. The menu is shown in the next table:

The number of observations per day contained in the synoptic data files is very different, depending on the station and the time. For example during the sixties most stations have recorded only two times per day, whereas in the eighties some stations have recorded hourly observations.

Therefore, the option REG/DAY allows to define which observations are used to create the climatological means. This is done by editing an integer array. Only observations at the hours given in the array are taken into account to calculate the means.

DAYS/MONTH allows to set a minimum number of days per month for which observations are required to calculate means. This comand carries a switch. If the switch is enabled only complete days are used according to the definition of REG/DAY.

26.3 VIEW: Display synoptical and climatological data

Using VIEW it is possible to display the synoptic and calculated climatological data. The menu VIEW has the form given in the table below. The comands SHOW, SCREEN and LIST_FILE are described in Sect. **??**.

SET-TIME and SELECT are the same options as they are described in the main menu.

Table 26.5:

VIEW (V1.0)	Displaying data and/or results
SET_TIME	Define time parameters for display
SELECT	Select kind of data to be displayed
SYNOP_DATA	Display all synop-data
METEOROLOGY	Display selected meteorological parameters
CLIMATE	Display selected climatological parameters
SCREEN	Set screen output to [Y/N]
LIST_FILE	Set list-file output to [Y/N]
INFO	Show information about menu

Table 26.6:

WRITE (V1.0)	Writing data to output files
SET_TIME	Define time parameters for output
SELECT	Select kind of data for output
METEOROLOGICAL	Write selected meteorological data to output file
STATION_CLIMATE	Write selected mean data to output file
APPEND_CLIMATE	Append data to climatological data file
INFO	Display information about menu

SYNOP_DATA displays all synoptical parameters exactly as they are read in from the input data file. This option may be used for control purposes.

METEOROLOGY shows the meteorological data of the selected parameters. A conversion to the standard units is performed.

Finally, CLIMATE allows to view the climatological means calculated for the selected parameters.

26.4   WRITE: Write data to output files

To create output files the WRITE menu has to be used. Its structure is rather similar to the VIEW menu, so the description may be kept shorter here. Especially the options SELECT and SET-TIME are the same and have the same effect on the output files as on the data shown on display. The menu is given in the next table:

Here, too, METEOROLOGY is used to write selected meteorological data to an output file. STATION-CLI. does the same for the selected monthly mean data. If no output data file was opened by calling FILES, the required files will be opened automatically. For more information about the default file names refer to FILES.

In a later version the output to an existing climatological data base will be possible by calling APPEND-CLI.

Chapter 27

ACIDE: Archive of comprehensive integrated data entities

ACIDE was designed to provide easy handling of a large number of time series. The time series were assumed to separated into subsets of different parameters. The analyses tools implemented in ACIDE provide utilities to related different parameters to each other.

The main menu of ACIDE is given in the next table.

27.1   WRITE: Write results and/or data to output files

!

Table 27.1:

ACIDE	Archive of comprehensive integrated data entities
FILES	Open files for I/O
CREATE	Create a new data base file
ADD_FILE	Add a file with standard db structure to contents
SET_TIME	Set time interval, origin, and units
SELECT_SERIES	Select series by parameters
FOR_ALL_SELECTS	Loop over all selected stations
NEXT_SELECT	For internal use together with FOR_ALL_SELECTS, only
ASSOCIATE	Associate stations of different parameters
DATA_MATRIX	Define and load data matrix
TRANSFORM	Transform the data in the data matrix
SAVE	Save data matrix to standard output file
ANALYSE	Analyse data or prepare for output
VIEW	View and/or print results and other parameters
WRITE	Write results to output file

Table 27.2:

WRITE	Write results to files
SEASONALITY	Save seasonal cycle to file
RESULTS	Save results of model fit to file
PARAMETERS	Write series parameters to file

Table 27.3:

FILES	Open input and output files
DATA_BASE	Open master data base file
ADD_FILE	Open a file to be added to the master file
MATRIX	Open output file to save the matrix
RESULTS	Output file to store results
SEASONALITY	Open output file to store seasonality
PARAMETERS	Open output file for series parameters

27.2   FILES: Open input and output files

27.3   EDISERI: Edit selection of time series

27.4   LOADAMA: Define, load or add data to matrix

27.5   ANALYSE: Define parameters for analyses and carry out analysis for selected

series

Table 27.4:

Name	Action
DELETE	
UNDELETE	
SELECT	
UNSELECT	

Table 27.5:

EDISERI	Edit selection of time series
ALL	Delete, undelete, select, unselect all series and types
TYPE_OF_SERIES	Act on specified type (given by name)
BY_NAME	Act on series specified by name
BY_INDEX	Act on series specified by a range of indexes
BY_AREA	Act on series specified by area
BY_COUNTRY	Act on series specified by country
DATA_TYPE	Select a specific data type (given by name)

Table 27.6:

LOADAMA	Define, load or add data to matrix
COLUMNS	Set number of columns in data matrix (including time)
INITIALIZE	Initialize data matrix and create time axis
SET_CONSTRAINTS	Define constraints for the columns
ASSIGN	Assign a series type to a column
LOAD	Load a series defined by station and type
RESET	Reset or initialize accumulator and counter
ACCUMULATE	Add given column to accumulator and counter

Table 27.7:

Time scale	Description
EQUIDISTANT	
MONTHLY	

Table 27.8:

Kind	Description
BY_INDEX	
BY_NAME	
NEAREST_TO	

Table 27.9:

ANALYSE	Define analysis parameter and analyse selected series
RESET	Delete any old results
DEFINE	Define general parameters
REGRESS	Calculate regression parameters
SET_MODEL	Set up model model for fit
FIT_MODEL	Fit model to data matrix
SEASONALITY	Determine seasonal cycle

Part V

The PRESENTation of data

Chapter 28

PLTEDIT: The graphical editor

The program PLTEDIT is an editor developed for designing graphical representation of data and texts.

Chapter 29

ISOLIN: Generation of isoline plots

Chapter 30

COLLOCATE: Construction of isolines by collocation

Chapter 31

GEOPLOT: Plotting geographic and thematic maps

Chapter 32

CONVER: Conversion of meta file for actual device-dependent output

Bibliography

Plag, H.-P., 1982. Analysis of tidal data from the Norwegian Coast., Tech. rep., IKU report P-203/1/82, Continental Shelf Institute, Trondheim.

Plag, H.-P., 1988. A regional study of Norwegian coastal long-period sea-level variations and their causes with special emphasis on the Pole Tide, *Berl. Geowiss. Abhandl. Reihe A*, **14**, 1–175.