Cora Kohlmeier, Frank Hamberg

MoViE 5.0.1

Model Visualization Environment

Oldenburg, June 25, 2025

Cora Kohlmeier
Institut für Chemie und Biologie des Meeres
Carl von Ossietzky Universität Oldenburg
c.kohlmeier@uol.de

Contents

1 Introduction 4

2 Installation of TigerPack 5

3 Main Controls 8

4 Overview Tool 12

5 Compare Tool 16

6 Film Tool 21

7 Text Tool 25

8 Profile Tool 28

9 Profile Compare Tool 32

10 Phase Tool 37

11 Contour Tool 41

12 [Analysis] 44

13 [Command] 47

14 Prepare 49

15 Options 52

16 Data structures 55

1 Introduction

The model visualization environment **MoViE** is part of **Tiger Graphics TigerPack** and was developed to simplify the model analysis of complex models. The philosophy of **MoViE** is to allow only reasonable actions and adapts itself to the loaded resultfiles. Thus, there is no need to remember f.e. the names of stored variables. **MoViE** is mainly adapted to the needs of box or layer models but it has been also successfully used for several kinds of problems (Hamberg, 1996).
For the model development **Tiger Graphics** provides **CEMoS**. A detailed description of using **CEMoS** and the implementation of complex models under **CEMoS** is given in (Hamberg, 1996; Hamberg und Kohlmeier, 2023).
MoViE is modular, thus, a lot of common functions are used by all tools. Nevertheless, those common functions are described with each tool to keep the description understandable and readable when only one tool is of actual interest.

License

This program is free software: you can redistribute it and/or modify it under the terms of the GNU General Public License as published by the Free Software Foundation, either version 3 of the License, or (at your option) any later version.

This program is distributed in the hope that it will be useful, but WITHOUT ANY WARRANTY; without even the implied warranty of MERCHANTABILITY or FITNESS FOR A PARTICULAR PURPOSE.

See the GNU General Public License for more details.

You should have received a copy of the GNU General Public License along with this program. If not, see http://www.gnu.org/licenses/.

2 Installation of TigerPack

Tiger Graphics TigerPack is written for UNIX or Linux[1]. It will also run on Mac OSX. Windows user may use it within a virtual machine.
TigerPack consists of 4 parts:

- **CEMoS** core system for the simulation
- **CEMTK** optional but highly recommended GUI for **CEMoS**
- **CEvoS** system for evolutionary parameter adaptation for **CEMoS** models
- **MoViE** visualization package

CEMTK and **CEvoS** can only be used if **CEMoS** is installed. **MoViE** can be used for any result files with appropriate format (see chapter 16).

2.1 Requirements

All part of **TigerPack** needs

- `csh`, `tcsh` or `bash`
- gcc-2.6.3 or higher
- `make`
- Tcl/TK[2]
- tkimg – the TKm image library for graphical output from **MoViE**

optional it is recommended to install

- `kwrite` or `gedit`
- `gdb` with `ddd` or `insight`
- `grace`
- `gv`
- `gnuplot`
- `kdiff3`

Nautilus/Dolphin users should also install

- `nautilus-open-terminal`

[1] Depending on the different properties of the operating systems the binary files are not portable from Intel machines to other architectures.

[2] **CEMoS** itself needs it only for the installation. This can also be done manually if Tcl/TK is not available

2.2 Installation

Copy the file `tigerpack.tar.gz` to the directory where do you like to install it and where do you have write permissions. Extract it by

```
tar -xzvf  tigerpack.tar.gz
```

Open a command terminal in the directory `TIGERPACK` and type the command

```
./setup
```

After that, logout and login again to set the environmental variables (reboot is NOT necessary).

2.3 Reinstalling after update

If you already have a running **TigerPack** and you would like to install an update Open a terminal in `TIGERPACK` and type the command `./reinstall`. No need to logout but existing **CEMoS** models must be recompiled.

2.4 Starting MoViE

MoViE can be started from any directory by `mov` It is convenient to start it from the directory where your simulation results are in. If a file `result.outc` already exists in the directory, it will be read, otherwise a dummy file `xxxx.outc` will be read. With **[Load Results]** own files can be read. The files must either be `.outc` files, `.nc` files or `.outa` files created with **CEMoS** files or files created with other programs with a suitable structure and extension. For the permitted data structures, see 16. On most systems it is also possible to start **MoViE** with a single or double click (depending on the system settings). In this case the clicked file will be loaded.

2.5 General operating instructions

MoViE needs for best performance all three mouse buttons. In the following chapters the term "middle mouse click" means to press the middle mouse button (resp. left, right). If not specified a left mouse click is meant.

For 3D plots (see 10) **MoViE** needs the package **Gnuplot** [3] is required. The current version of **MoViE** is working with **Gnuplot 3.7 patchlevel 3** . If **Gnuplot** is not

[3] **Gnuplot** is part of Linux distributions.
Gnuplot can be obtained from http://www.gnuplot.info.

available, only 2D plot are possible in the phase tool.

Field data that shall be visualized must be stored in a predefined format (see 16). The main restriction is that all data must be stored as one dimensional indexed arrays.

3 Main Controls

3.1 MoViE tools

The available tools are selected via the menu at the top left

- **Overview:** 2D time plots of one or more variables and boxes (4)
- **Compare:** comparing of different simulation results and comparing of results with measured data (5)
- **Film:** animation of a selected variable within some predefined maps (6)
- **Text:** comparing of different simulation results via numerical output (7)
- **Profile:** profile plot where the values are plotted over all boxes resp. layers for a specific time step (8).
- **ProfCom:** profile plot with the capability of comparing different files and displaying measured data (9).
- **Phase:** 2D phase plots of 2 selected variables (described in chapter 10)
- **Contour:** Contour plot. Displaying values for all time steps and boxes resp. layers. The values are encoded by colors (11).

3.2 The variable selector – all tools

The variable selector contains a button for every variable stored in the selected result file (see figure 4.1). A variable is selected by clicking the button with the left mouse button. The default variable is the variable on the first button. If a variable is selected which has not been stored for the given box a message appears in the info window. The color for a button can be set in the file `name.info` (see 16.7). In this file additional informations for the variable -such as units- which is displayed in the plot can be set.
An alternative variable selector (nice selector) which contains other names instead of variable names can be constructed. By default a file named `nicevar.info` is read if available (see 16.8).

3.3 The box selector – Overview, Compare, Text, Phase

The box selector contains a button for every box stored in the selected result file (see figure 4.1) . A box is selected by clicking the button with the left mouse button. The default box is the box on the first button. If a variable-box combination is selected

which has not been stored in the result file a message appears in the info window. If a box is selected which has not been stored for the given variable a message appears in the info window .

3.4 The Time slider – Profile, Profcom

The actual time for the plot can be modified by the time slider.
A range in time can be input manually in the text areas below the time slider. The values become active, after pressing ENTER. The values in these text areas are delimiting the range in time for the sequence Postscript outputs, which can be produced by pressing the button **[Record to Postscript]** .

Remark: Every time the slider is set to the start time (left) the results will be extracted for the complete time interval. This may take some time.

3.5 Load Results

[Load Results] – Overview, Film, Profile, Phase, Contour

A file selector window appears which shows all result files in the working directory. Valid result file are binary files from **CEMoS** (`.outc` files), **NetCDF** files (`.nc` files, for correct notation see **CEMoS** and **NetCDF** manuals) or comma seprated ASCII files written by **CEMoS** or any other program (`.outa` files, for data structure see 16).
A result file can be selected by left mouse click and confirmed by **[OK]** . The loading procedure will take some time depending on the size of the result file. After loading, the file selector box disappears and a message appears in the info window.
_pk.outc containing spectrum data (made by **MoViE**) may be also loaded. **MoViE** recognizes automatically if the file contains Fourier coefficients instead of simulation results and directs it into a Fourier synthesis routine.
If more than one file is loaded only the common variables, boxes and time window will be displayed.
Remark: If a file has already been loaded within another tool it will be reloaded in the actual tool.

[Load Results] – Compare, Text, Profcom

A file select window appears which contains buttons for loading up to five result files. At start only the first one is loaded, the following are empty. If more than one result file is loaded the following files may be released with **[Release all]** .
[Cancel] closes the window. Clicking on the buttons opens a file selector window which shows all result files in the working directory. A result file can be selected by mouse click and confirmed by **[OK]** . The loading procedure will take some time

depending on the size of the result file. After loading, the file selector box and the file selector window disappear and a message appears in the info window.

If further results files shall be loaded **[Load Results]** has to be pressed again.
`_pk.outc` may be also loaded . **MoViE** recognizes automatically if the file contains Fourier coefficient instead of simulation results and directs it into a Fourier synthesis routine.
If more than one file is loaded only the common set of variables, boxes and the commom time window will be displayed.
Remark: If files have already been loaded within another tool they will be reloaded in the actual tool.

3.6 [Load Data] – only Compare, Profcom

A file select window appears which contains buttons for loading data files. After pressing the button a selector window appears which shows all data files (`.symb` files) in the default validation directory (see 15) or in the working directory if no default is set.
A file can be selected by mouse click and confirmed by **[OK]** .
If further data files shall be loaded **[Load Results]** has to be pressed again.

3.7 Map

This is a special function for models based on the different ERSEM setups. For models whose number of boxes corresponds to that of the setup, the respective map is selected and can be displayed here.

3.8 Export

[PDF] ,[PostScript] , [xmGrace]

The output is exported as .pdf, .ps , .agr or .txt file.
The file is named `<toolname>_<date>_<time>.<outputtype>` by default and is stored in the directory `<working directory>/plots`.
The available export formats depend on the current tool.

3.9 Export

[text] (only text tool

The output is exported as .txt file. The output is saved as shown in the output window. If a comma separated output is desired, the result file can be converted into

an .outa file (see 14). The file is named `text_<date>_<time>.txt` by default and is stored in the directory `<working directory>/plots`.

[MatLab]

A MatLab[1] script file is created which reads the data for every graph and set from separate files. All files are stored in the directory `<working directory>/plots/matlab_<date>_<time>` by default. The script file is named `matlab_<date_time>.m`. The data files get the same name which is extended by graph number y and set number z `matlab_<date>_<time>_y_z.dat`.

[Gnuplot]

A GNUPLOT[2] batch file is created which reads the data for every graph and set from separate files. All files are stored in the directory `<working directory>/plots/gnuplot_<date>_<` by default. The script file is named `matlab_<date_time>.m`. The data files get the same name which is extended by graph number y and set number z `gnuplot_<date>_<time>_y_z.dat`.

3.10 Box Slider

Instead of buttons, the boxes can be selected with a slider.

3.11 Nice Varselector

An alternative variable selector (nice variable selector) which contains other names instead of the model variable names can be constructed. For this a file named `nicevar.info` must be available in the directory where **MoViE** has been started. For the data structure of the file `nicevar.info` see 16.8 .

3.12 Single Window mode

If checked he control panel, the variable selector and the box selector are undocked.

[Exit]

Closes **MoViE**.

[1] MATLAB, The MathWorks, Inc., Natick, Massachusetts, United States.
[2] http://gnuplot.sourceforge.net/

The Info Window

All informations and errors will be displayed here.

4 Overview Tool

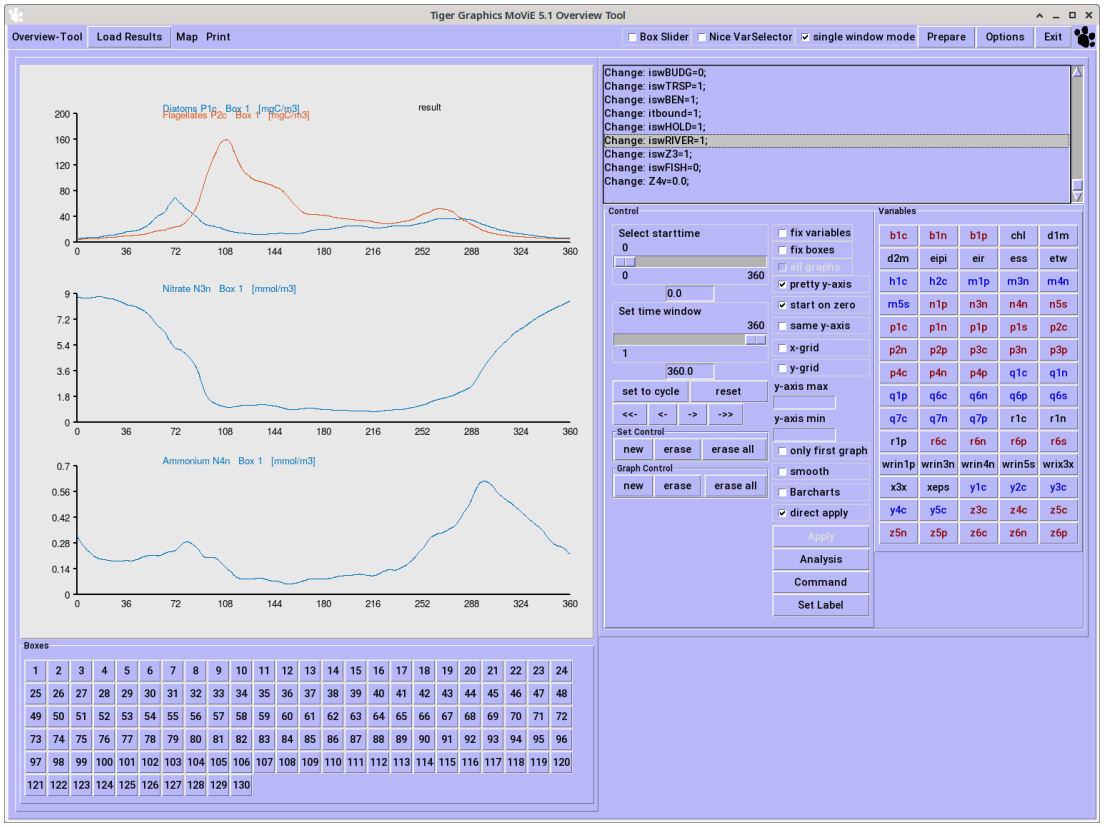

Figure 4.1: The overview tool

The overview tool allows a close look to the change in time of every stored variable in each box. It is possible to have a look on sets of variables in up to nine graphs. These variables might be of one box or of different boxes. After selecting a set of variables (boxes) these can be fixed and can be shown for different boxes (variables). Starttime and a time window can be selected to show only temporal parts of the simulation results.
Time plots are displayed for the selected variables in the selected boxes. The axis labels of the time axis are set automatically in respect to the information within the result file and the stored time interval.

By means of the file `name.info` (see 16.7) variable names from simulations may be translated to extended names with units. **MoViE** will read this file and the full name and the unit of the variable will be displayed if available and reasonable. If no information about a variable is given in `name.info` the name stored in the simulation result file will be used.

Overview control panel

The control panel contains all buttons and sliders for modifying the actual plots and the plotting mode.

[Select starttime] and [Set time window]

The starting time of the plot and the period which shall be plotted can be selected. If the time window exceeds the endtime for a starttime the time window will be reduced accordingly.

[set to cycle] and [reset]

The time window is set to the value given by the variable cycle (default 360). If the variable cycle is set in the simulation parameters of the result file (see cem manual), this value is read. Reset reetsets the time window to the maximum length. **[<]** , **[>]** allows stepping cycle by cycle, **[<<]** ,**[>>]** jumps to the first resp. last cycle.

[new set]

This button activates the next plot line (set). The same action can be activated by a middle mouse click on the variable/box.
This new set remains active for all actions (f.e. from **[analysis]**) until

- another set is activated (**[new set]** or left mouse click onto a variable/box)
- it is erased (**[erase set]** or right mouse click onto the set)
- a new graph is activated (**[new graph]** or right mouse click onto a variable/box)
- the actual graph is erased **[erase graph]**
- all graphs are erased **[erase all graphs]**

You may have up to 9 sets per graph.

[erase set]

This button erases the actual (the last) set from the actual graph. After this the "new" last set of the actual graph becomes active again.

[erase all]

This button erases the all sets from the actual graph. After this the "new" last set of the actual graph becomes active again.

[new graph]

This button activates a new graph. This button is redundant. The same action can be activated by a right mouse click on the variable/box.
You may have up to nine graphs on in the drawing area, which will be automatically arranged.

[erase graph]

This button erases the actual graph with all its sets. The "new" last graph becomes active again. This button becomes active after pressing **[new graph]** or if two or more graphs are displayed in the drawing area. It becomes inactive after pressing **[erase all graphs]** or if only one active graph remains in the drawing area.

[erase all graphs]

This button erases all graphs with all their sets. This is useful to clear up the drawing area if you are loosing the overview.

[fix variables]

The variable selector will become inactive and the present selection of variables will be fixed for the next actions. After selecting a new box the plot will display the fixed variables for the selected box. Clicking again on **[fix variables]** deactivates this mode and the variable selector becomes active again.
If **[new graph]** is pressed whilst **[fix variables]** is active, all information is piped from the previous graph to the new one except the box numbers which are replaced by the next selected one.

[fix boxes]

The box selector will become inactive and the present selection of boxes will be fixed for the next actions. After selecting a new variable the plot will display the selected variable for the fixed boxes. Clicking again on **[fix boxes]** deactivates this mode and the box selector becomes active again.
If **[new graph]** is pressed whilst **[fix boxes]** is active, all information is piped from the previous graph to the new one except the variable name which are replaced by the next selected one.

[act on all graphs]

This button becomes active if **[fix boxes]** or **[fix variables]** is selected.
If **[act on all graphs]** is pressed the fixes in variables or boxes will act on all displayed graphs. Pressing **[act on all graphs]** again deactivates this option. The fixes will work on the last (the actual) graph only again.

[pretty y-axis]

After pressing **[pretty y-axis]** all graphs in the drawing area get a scaling on the y-axis that matches to round (nice) numbers. Such, if the maximum of the displayed variable is f.e `8.41` activating **[pretty y-axis]** results in a scaling to an axis of `9.0`. Otherwise, there will be an exact scaling to `8.41`.

[start on zero]

The y-axis starts on zero even if there are negative data

[same y-axis]

After pressing **[same y-axis]** all graphs in the drawing area get the same scaling on the y-axis. This is useful if the same variable is drawn for different boxes. The global scaling stays active until **[same y-axis]** is pressed.

[x-grid]

When clicked, grid lines on x-values are displayed. The spacing of the lines is controlled by the variable cycle (see Hamberg und Kohlmeier (2023)) if set , otherwise The spacing of the lines is controlled by the displayed tick marks.

[y-grid]

When clicked, grid lines on y-values are displayed. The spacing of the lines is controlled by the displayed tick marks.

[y axis maximum] [y axis minimum]

If only a limited range of values shall be displayed, this range can be input manually in the text areas below the labels. The values become active, after pressing `ENTER` .
If no values are filled in the maximum resp. minimum from the data to be displayed will be taken into account.
If **[only first graph]** is active, the settings of minimum and maximum will only take effect on the first graph.

[only first graph]

When clicked **[y axis maximum]** and **[y axis minimum]** are evaluated for the frist graph only.

[smooth]

If **[smooth]** is active, the line segments are rendered with a set of Bezier splines otherwise they will be displayed as a polygon.

[Barchart]

The data will be displayed as barcharts instead of lines.

[direct apply]

If **[direct apply]** is active, all clicks in the box selector or the variable selector, as well, as all actions on the different sliders will be immediately evaluated.
If **[direct apply]** is not active, the actual settings (also creation of new sets or graphs) will be evaluated when the button **[Apply]** is pressed. Such, it is possible to prepare more complex displays in a kind of 'offline' mode and get the results displayed in one evaluation.

[Analysis]

See chapter 12.

[Command]

See chapter 13

[Set Label]

Additional text can be added to the plot in different fontsizes. The text must be given in the text field. After confirming this by ENTER the text appears in the drawing area. It can be moved by pressing the left mouse button on it and moving the mouse holding the button pressed. After releasing the mouse button the title jumps to that position. The label can be removed by a right mouse click while holding the shift button pressed.
The label will only appear in the PostScript resp. pdf outputs!

5 Compare Tool

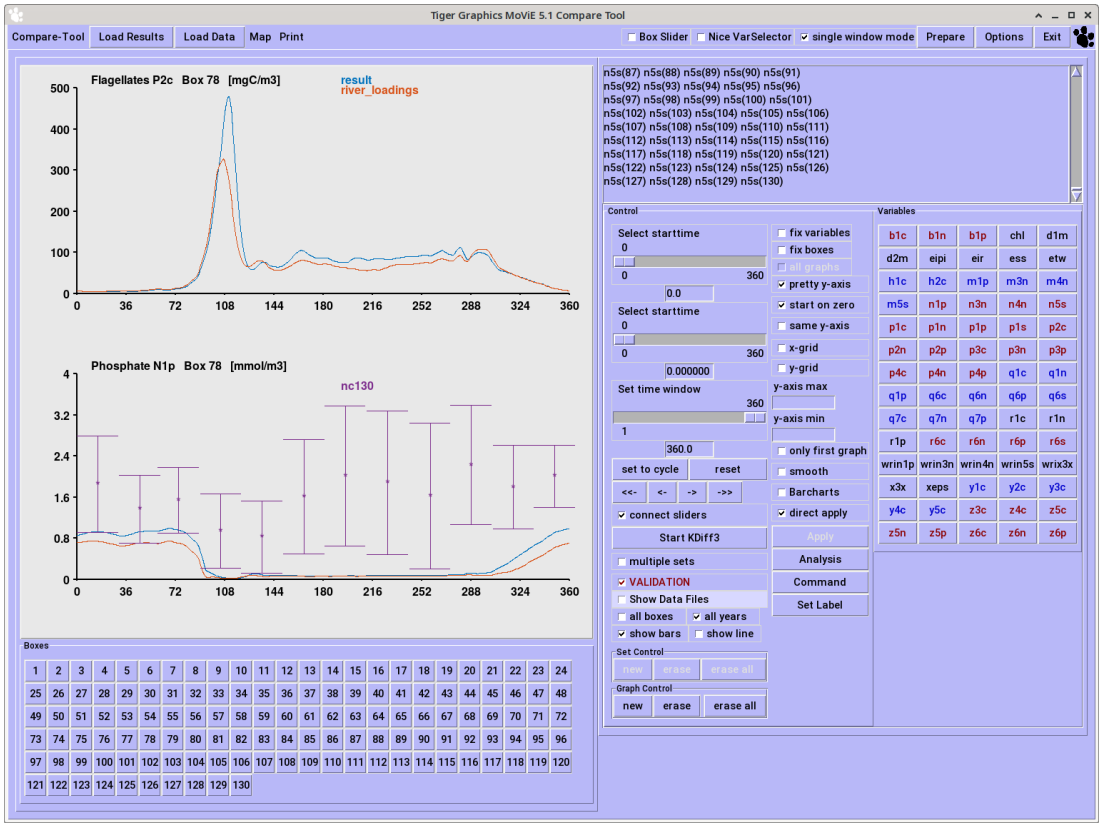

Figure 5.1: Compare tool

This tool allows a comparison of different result files and the comparison of model results and field measurements (For the format of these data sets see appendix 16). The compare tool allows also the comparison of different periods of f.e. the same result file to investigate the temporal changes during simulation.

Compare control panel

The control panel contains all buttons and sliders for modifying the actual plot and the plotting mode.

[Select starttime] and [Set time window]

The starting time of the plot can be selected independently for every result file with **[Select starttime 1]** ,**[Select starttime 2]** and **[Select starttime 3]** . The time window can be set with the common **[Set time window]** slider. If the time window

exceeds the endtime for a starttime the time window will be reduced. The starttime can be set for all result files in common by **[connect sliders]** .

[connect sliders] /[disconnect sliders]

If the starttime for all graphs shall be the same the **[connect sliders]** shall be pressed. Now all graphs get the starttime given by **[Select starttime 1]** . The other starttime sliders will be deactivated and the name of the button changes to **[disconnect sliders]** . The connect mode is now active until the button **[disconnect sliders]** is pressed. This is the configuration at start of the tool.

[multiple sets]

If activated more than one set can be selected. If only one result file is loaded, the compare tool behaves similar to the overview tool. If more than one result file is loaded the sets for the different results are distinguished by different line thicknesses while the different sets are distinguished by different colors. This may be sometimes a little bit confusing. Activating **[multiple sets]** activates the controls for **[new set]** ,**[erase set]** and **[erase all sets]**

[validation]

Accessible if field data are loaded (see 3.6). If more than one data set is loaded a window opens where every data set may be activated or deactivated. By default is all loaded data sets are activated. The activated sets are shown together with the model results in the plot window if measured data are available for the selected variable/box combination. The data must have the structure defined in 16.5.
MoViE doesn't distinguish data for different years.

[number of harmonics]

If one loaded result file is a pk-file (16.2) this button appears. The number of harmonics can be selected where 0 means that for all loaded pk-files all available harmonics are evaluated (default). The maximum number which can be selected manually is the smallest number of harmonics available for all loaded pk-files.

[new set]

This button activates the next plot line (set). The same action can be activated by a middle mouse click on the variable/box.
This new set remains active for all actions (f.e. from **[analysis]**) until

- another set is activated (**[new set]** or left mouse click onto a variable/box)
- it is erased (**[erase set]** or right mouse click onto the set)

- a new graph is activated (**[new graph]** or right mouse click onto a variable/box)
- the actual graph is erased **[erase graph]**
- all graphs are erased **[erase all graphs]**

You may have up to 9 sets per graph.

[erase set]

This button erases the actual (the last) set from the actual graph. After this the "new" last set of the actual graph becomes active again.

[erase all]

This button erases the all sets from the actual graph. After this the "new" last set of the actual graph becomes active again.

[new graph]

This button activates a new graph. This button is redundant. The same action can be activated by a right mouse click on the variable/box.
You may have up to nine graphs on in the drawing area, which will be automatically arranged.

[erase graph]

This button erases the actual graph with all its sets. The "new" last graph becomes active again. This button becomes active after pressing **[new graph]** or if two or more graphs are displayed in the drawing area. It becomes inactive after pressing **[erase all graphs]** or if only one active graph remains in the drawing area.

[erase all graphs]

This button erases all graphs with all their sets. This is useful to clear up the drawing area if you are loosing the overview.

[fix variables]

The variable selector will become inactive and the present selection of variables will be fixed for the next actions. After selecting a new box the plot will display the fixed variables for the selected box. Clicking again on **[fix variables]** deactivates this mode and the variable selector becomes active again.
If **[new graph]** is pressed whilst **[fix variables]** is active, all information is piped from the previous graph to the new one except the box numbers which are replaced by the next selected one.

[fix boxes]

The box selector will become inactive and the present selection of boxes will be fixed for the next actions. After selecting a new variable the plot will display the selected variable for the fixed boxes. Clicking again on **[fix boxes]** deactivates this mode and the box selector becomes active again.
If **[new graph]** is pressed whilst **[fix boxes]** is active, all information is piped from the previous graph to the new one except the variable name which are replaced by the next selected one.

[act on all graphs]

This button becomes active if **[fix boxes]** or **[fix variables]** is selected.
If **[act on all graphs]** is pressed the fixes in variables or boxes will act on all displayed graphs. Pressing **[act on all graphs]** again deactivates this option. The fixes will work on the last (the actual) graph only again.

[pretty y-axis]

After pressing **[pretty y-axis]** all graphs in the drawing area get a scaling on the y-axis that matches to round (nice) numbers. Such, if the maximum of the displayed variable is f.e `8.41` activating **[pretty y-axis]** results in a scaling to an axis of `9.0`. Otherwise, there will be an exact scaling to `8.41`.

[start on zero]

The y-axis starts on zero even if there are negative data

[same y-axis]

After pressing **[same y-axis]** all graphs in the drawing area get the same scaling on the y-axis. This is useful if the same variable is drawn for different boxes. The global scaling stays active until **[same y-axis]** is pressed.

[y axis maximum] [y axis minimum]

If only a limited range of values shall be displayed, this range can be input manually in the text areas below the labels. The values become active, after pressing `ENTER` .
If no values are filled in the maximum resp. minimum from the data to be displayed will be taken into account.
If **[only first graph]** is active, the settings of minimum and maximum will only take effect on the first graph.

[only first graph]

When clicked **[y axis maximum]** and **[y axis minimum]** are evaluated for the frist graph only.

[smooth]

If **[smooth]** is active, the line segments are rendered with a set of Bezier splines otherwise they will be displayed as a polygon.

[Barchart]

The data will be displayed as barcharts instead of lines.

[direct apply]

If **[direct apply]** is active, all clicks in the box selector or the variable selector, as well, as all actions on the different sliders will be immediately evaluated.
If **[direct apply]** is not active, the actual settings (also creation of new sets or graphs) will be evaluated when the button **[Apply]** is pressed. Such, it is possible to prepare more complex displays in a kind of 'offline' mode and get the results displayed in one evaluation.

[Analysis]

See chapter 12.

[Command]

See chapter 13

[Set Label]

Additional text can be added to the plot in different fontsizes. The text must be given in the text field. After confirming this by `ENTER` the text appears in the drawing area. It can be moved by pressing the left mouse button on it and moving the mouse holding the button pressed. After releasing the mouse button the title jumps to that position. The label can be removed by a right mouse click while holding the shift button pressed.
The label will only appear in the PostScript resp. pdf outputs!

6 Film Tool

This tool needs a click to **[Apply]** by default!

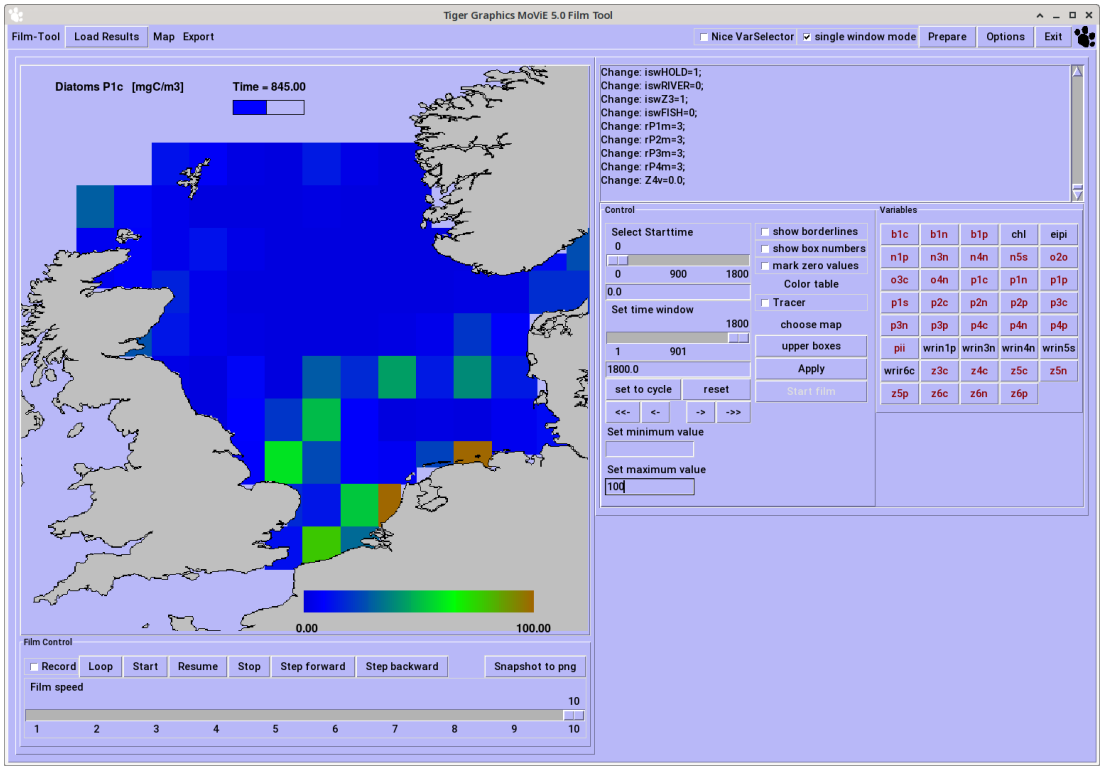

Figure 6.1: Film animation tool

This tool allows an animation of a selected variable in a predefined map.

The animation works highly interactive: single pictures, different start and end time, different color tables and different scalings are possible.

The definition of a new map requires knowledge in programming **Tcl/TK** . The maps have to be adapted to the specific models.

The capability of the film tool can be seen by loading the default `result.outc` file included in **MoViE** . It is an output file from the 15-box model version of ERSEM.

Remark: This file is automatically loaded if the working directory does not contain a file named `result.outc`

6.1 Film control panel

The control panel contains all buttons and sliders for preparing the animation.

[Select starttime] and [Set time window]

The starting time of the plot and the period which shall be plotted can be selected. If the time window exceeds the endtime for a starttime the time window will be reduced accordingly.

[set to cycle] and [reset]

The time window is set to the value given by the variable cycle (default 360). If the variable cycle is set in the simulation parameters of the result file (see cem manual), this value is read. Reset reetsets the time window to the maximum length. **[<]** , **[>]** allows stepping cycle by cycle, **[<<]** ,**[>>]** jumps to the first resp. last cycle.

[Upper Boxes] /[Lower Boxes]

The name of the button shows the selected layer of boxes. Due to the box set up surface or sediment boxes can be animated. The button activates **[apply]** .

[Set minimum] and [Set maximum]

These buttons affect the color scaling. By default the minimum and maximum values are taken over the whole stored simulation time for the selected boxes scheme (lower or upper). In the text fields the minimum and maximum values may be set by hand. All values above the maximum (below the minimum) are drawn in the "maximum" ("minimum") color.
The use of this is given if f.e. the abundance of phytoplankton during winter shall be investigated. The calculation of the maximum over a whole year leads to a very high dynamics during the spring bloom. Such there will be no color contrast during winter.
These buttons activate **[apply]** .

[show borderlines]

If activated border lines are displayed during animation.

[show boxnumbers]

If activated box numbers are displayed after animation.

[mark zero values]

If activated zeros are displayed in grey and not in a color of the color table.

[Color table]

opens a menu with different color tables which can be selected. This must be confirmed by **[apply]** !!

[tracer]

Remark: Only valid for ECOTIM. If an EulerBox variable is selected and the corresponding values for all voxels are stored including the `xcoor` and `ycoor` variables, the tracers are animated.

[Choose map]

Normally **MoViE** loads the correct setup automatically. Here it can be set manually. This must be confirmed by **[apply]** !!

[upper boxes]

!! **Remark:** Only valid for ERSEM like models whre upper and lower boxes are stored (see Hamberg und Kohlmeier (2023).

[apply]

The map, the boxes and the selected variable at the selected starttime will be drawn. The color bar shows the range of values and the corresponding color.
After every modification which affects the actual selection such as changes in the selected variable, starting time, lower or upper boxes and re-scalings this button becomes active and must be pressed before the next animation. It takes care that the given colors and color ranges, names and units correspond to the actual choice. This avoid unnecessary data extractions and saves time when working on large simulation results.
After data extraction and preparation for the animation the button **[Start film]** is activated.
Remark: The extraction of data takes some more time than in the other tools, because data must be extracted for all displayed boxes.

Remark: MoViE makes no check if a variable is stored for all boxes. Errors still may occur.
Remark: The name.info file (16.7) will be read. The full name and the unit of the variable will be printed as title.

[Film Control]

[Record] If activated every image is saved as png file in a directory in the directory `<working directory>/film`. The directory for the png-files is named by date

and time, the png files are named `frame_x.png` and are numbered consecutively. Recording slows down the animation. It must be ensured that sufficient storage space is available. Each png has a size of about 50 kByte. The window is kept in front during recording. The png files can be combined to a video f.e. by `convert *.png all.mpg`, if convert is installed.If activated every image is saved as separate png-file in `<modeldir>/films`.

[Start film] opens the film control.

[Loop] starts the animation and the film will be repeated until **[Stop]** is pressed.

[Start] starts the animation. The animation stops at the end of the time window.

[Resume] continues the animation. The animation stops at the end of the time window.

[Stop] stops the animation at the actual time.

The **[Step forward]** and **[Step backward]** buttons allow single step animations within the selected time window.

[Snapshot to png] saves the actual image as png file as `filmsnapshot_<datetime>.png` in the directory `<working directory>/plots`.

The slider **[Film speed]** allows a slow motion animation.

7 Text Tool

This tool provides a view on the numerical values of all results in up to three result files. It is built for mass conservation checks and program checking. It allows adding of variables, multiplying variables by a scalar and a brief statistical analysis.

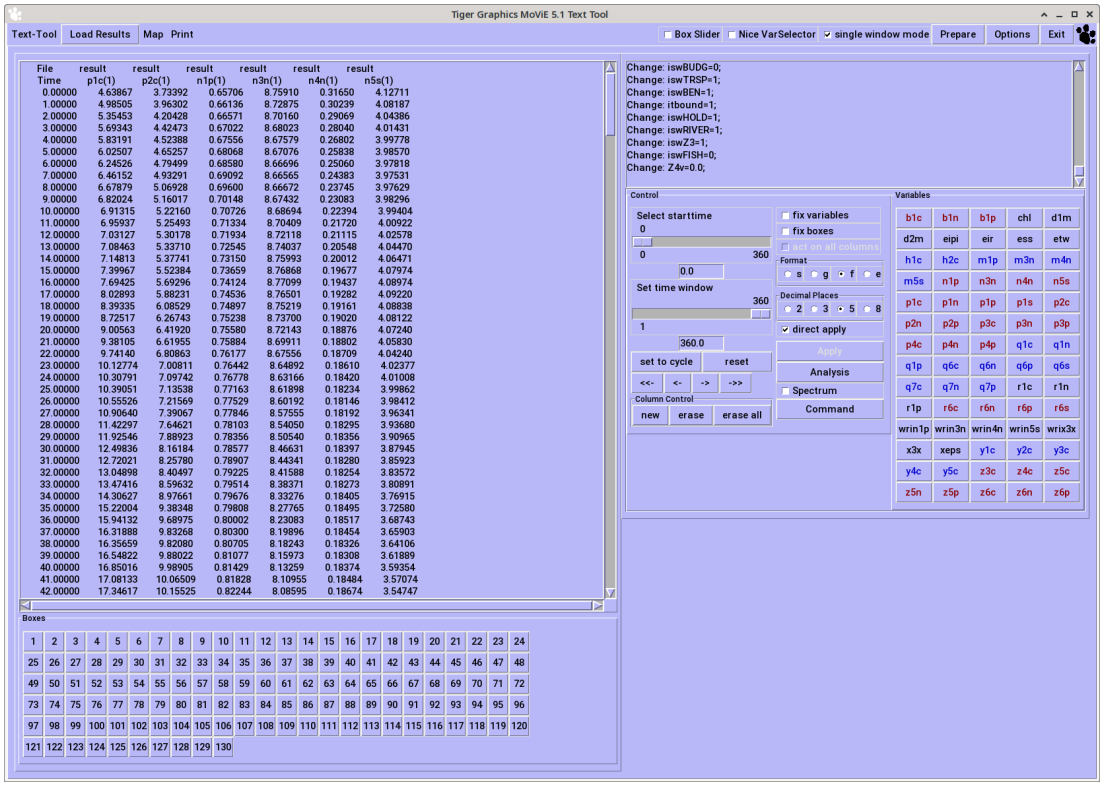

Figure 7.1: Text tool

Text Area

The text area is a scrollable window where the numerical output is printed column by column. It is possible to mark text in it and copy it to other applications etc.

Text control panel

The control panel contains all buttons and sliders for modifying the numerical output.

[Select starttime] and [Set time window]

The starting time of the plot and the period which shall be plotted can be selected. If the time window exceeds the endtime for a starttime the time window will be reduced accordingly.

[set to cycle] and [reset]

The time window is set to the value given by the variable cycle (default 360). If the variable cycle is set in the simulation parameters of the result file (see cem manual),

this value is read. Reset reetsets the time window to the maximum length. **[<]** , **[>]** allows stepping cycle by cycle, **[<<]** ,**[>>]** jumps to the first resp. last cycle.

[new column]

The next selected variable will be printed in an additional column. Here a column must be understood as a block of x columns where x denotes the number of loaded files. The same action can be activated by a right mouse click on the variable/box.
Remark: The blocks of columns are structural identical to the graphs in the compare tool! In the following a column means a block of columns if not specified differently!
All modifications as f.e. changing the variable will be done in this column (block) until **[new column]** is pressed again. There is an internal limitation of 9 columns (blocks).

[erase column]

Only active if two or more columns are printed.
Deletes successively the last column. The next operations will affect the "new" last column.

[erase all]

Only active if two or more columns are printed. All displayed columns will be deleted.

[fix variables]

The variable selector will become inactive and the present selection of variables will be fixed for the next actions. After selecting a new box the plot will display the fixed variables for the selected box. Clicking again on **[fix variables]** deactivates this mode and the variable selector becomes active again.
If **[new column]** is pressed whilst **[fix variables]** is active, all information is piped from the previous column to the new one except the box numbers which are replaced by the next selected one.

[fix boxes]

The box selector will become inactive and the present selection of boxes will be fixed for the next actions. After selecting a new variable the plot will display the selected variable for the fixed boxes. Clicking again on **[fix boxes]** deactivates this mode and the box selector becomes active again.
If **[new column]** is pressed whilst **[fix boxes]** is active, all information is piped from the previous column to the new one except the variable name which are replaced by the next selected one.

[act on all columns]

This button becomes active if **[fix boxes]** or **[fix variables]** is selected.
If **[act on all columns]** is pressed the fixes in variables or boxes will act on all displayed columns. Pressing **[act on all columns]** again deactivates this option. The fixes will work on the last (the actual) column only again.

[Format]

The number representation can be set to the common numeric formats. This only affects the following selections.

[Digits]

The number of digits can be set. This only affects the following selections. If all stored digits are desired selct string format (s).

[direct apply]

If **[direct apply]** is active, all clicks in the box selector or the variable selector, as well, as all actions on the different sliders will be immediately evaluated.
If **[direct apply]** is not active, the actual settings (also creation of new columns) will be evaluated when the button **[Apply]** is pressed. Such, it is possible to prepare more complex displays in a kind of 'offline' mode and get the results displayed in one evaluation.

[Analysis]

See chapter 12.

[Spectrum]

Only valid if a Fourier result file is loaded. In this case the spectral values are shown.

[Command]

See chapter 13

Figure 8.1: Profile tool

8 Profile Tool

This tool allows a close look to the change of concentration in different layers. Here the typical box structure is interpreted as layer structure. Every variable must be stored for a continuous set of layers (boxes).
The concentration is displayed on the x-axis for every layer starting on top with the layer with the smallest number.
If a variable named `layer` is stored for a continuous set of box numbers those values are interpreted as thickness of the corresponding layer.These layer information may vary in time. So dynamically adapted layer thicknesses can be visualized. The sets are scaled corresponding to these depths. If the variable `layer` is not stored the results are plotted equidistantly.
General features of the profile tool

- up to nine graphs simultaneously (not recommended [1])
- up to nine variables in one graph
- variation of the simulation time with the time slider

[1] The whole data set for all shown layers and all time steps are read if a variable is selected. This may take some time especially if only limited memory is available

The title of the plot shows the names of the variables and the actual time. If a file name.info exists it will be read. The full name and the unit of the variable will be displayed if available.

Profile control panel

The control panel contains all buttons and sliders for modifying the actual plot and the plotting mode.

[Select startbox] and [Set box window]

The first displayed box and the numbers of box to follow can be selected.
If the time window exceeds the maximum number of boxes for startbox the box window will be reduced internally.

[new set]

This button activates the next plot line (set). The same action can be activated by a middle mouse click on the variable/box.
This new set remains active for all actions (f.e. from **[analysis]**) until

- another set is activated (**[new set]** or left mouse click onto a variable/box)
- it is erased (**[erase set]** or right mouse click onto the set)
- a new graph is activated (**[new graph]** or right mouse click onto a variable/box)
- the actual graph is erased **[erase graph]**
- all graphs are erased **[erase all graphs]**

You may have up to 9 sets per graph.

[erase set]

This button erases the actual (the last) set from the actual graph. After this the "new" last set of the actual graph becomes active again.

[erase all]

This button erases the all sets from the actual graph. After this the "new" last set of the actual graph becomes active again.

[new graph]

This button activates a new graph. This button is redundant. The same action can be activated by a right mouse click on the variable/box.
You may have up to nine graphs on in the drawing area, which will be automatically arranged.

[erase graph]

This button erases the actual graph with all its sets. The "new" last graph becomes active again. This button becomes active after pressing **[new graph]** or if two or more graphs are displayed in the drawing area. It becomes inactive after pressing **[erase all graphs]** or if only one active graph remains in the drawing area.

[erase all graphs]

This button erases all graphs with all their sets. This is useful to clear up the drawing area if you are loosing the overview.

[eval layer]

If **[eval layer]** is selected and if the result file contains information on the layer thickness (stored in the variable `layer`), the information on time changing layer thicknesses are evaluated to compose the plot of the profile. If the variable `layer` is not stored or if **[eval layer]** is not activated all layer thicknesses are assumed to be equal and constant.

[show layer value]

If **[show layer value]** is selected and if the result file contains information on the layer thickness (stored in the variable `layer`), the information on time changing layer thicknesses are evaluated and the depth (starting from top) is displayed on the layer axis instead of the layer number.

[flip x-y]

Change to a plot with exchanged concentration and layer axes.

[same x-axis]

If this option is active all graphs in the drawing area get the same scaling on the x-axis. This is useful if the same variable is drawn for different boxes.

[scale to time maximum]

If this option is active the x-axis maximum is set to the maximum over the whole time interval.

[Barchart]

The data will be displayed as barcharts instead of lines.

[smooth]

If **[smooth]** is active, the line segments are rendered with a set of Bezier splines otherwise they will be displayed as a polygon.

[direct apply]

If **[direct apply]** is active, all clicks in the box selector or the variable selector, as well, as all actions on the different sliders will be immediately evaluated.
If **[direct apply]** is not active, the actual settings (also creation of new sets or graphs) will be evaluated when the button **[Apply]** is pressed. Such, it is possible to prepare more complex displays in a kind of 'offline' mode and get the results displayed in one evaluation.

[Analysis]

See chapter 12.

[Command]

See chapter 13

[Set Label]

Additional text can be added to the plot in different fontsizes. The text must be given in the text field. After confirming this by ENTER the text appears in the drawing area. It can be moved by pressing the left mouse button on it and moving the mouse holding the button pressed. After releasing the mouse button the title jumps to that position. The label can be removed by a right mouse click while holding the shift button pressed.
The label will only appear in the PostScript resp. pdf outputs!

9 Profile Compare Tool

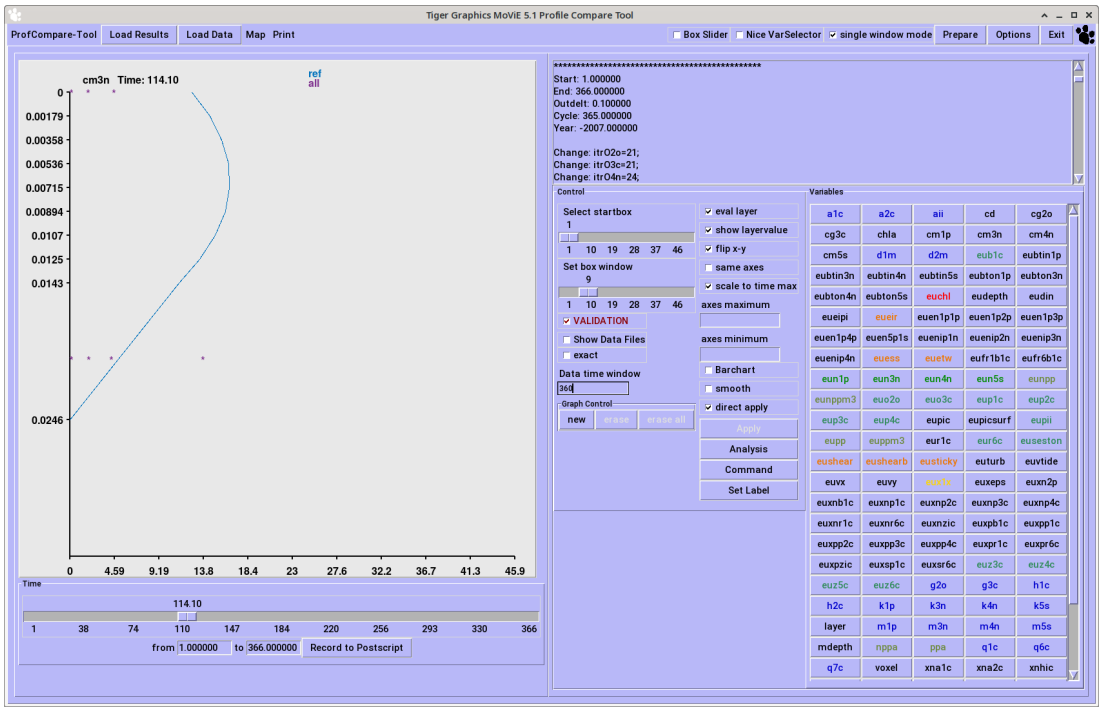

Figure 9.1: Profile compare tool

This tool allows a close look to the change of concentration in different layers. It combines the capabilities of the Compare Tool (see 5) and the Profile Tool (see 8). Extended use of measured data is possible by a lot of control functions. Here the typical box structure is interpreted as layer structure. Every variable must be stored for a continuous set of layers (boxes).
The concentration is displayed on the x-axis for every layer starting on top with the layer with the smallest number.
If a variable named `layer` is stored for a continuous set of box numbers those values are interpreted as thickness of the corresponding layer.These layer information may vary in time. So dynamically adapted layer thicknesses can be visualized. The sets are scaled corresponding to these depths. If the variable `layer` is not stored the results are plotted equidistantly.
General features of the profile compare tool

- up to nine graphs simultaneously (not recommended [1])
- variation of the simulation time with the time slider

[1] The whole data set for all shown layers and all time steps are read if a variable is selected. This may take some time especially if only limited memory is available

The title of the plot shows the names of the variables and the actual time. If a file name.info exists it will be read. The full name and the unit of the variable will be displayed if available.

Profcom control panel

[Select startbox] and [Set box window]

The first displayed box and the numbers of box to follow can be selected.
If the time window exceeds the maximum number of boxes for startbox the box window will be reduced internally.

[validation]

Accessible if field data are loaded (see 3.6). If more than one data set is loaded a window opens where every data set may be activated or deactivated. By default is all loaded data sets are activated. The activated sets are shown together with the model results in the plot window if measured data are available for the selected variable/box combination. The data must have the structure defined in 16.5.
MoViE doesn't distinguish data for different years.

[exact]

When activated and if **[validation]** is active, measured data are only displayed, if they are available for the exact point in time given by the time slider. By default **[Exact]** is deactivated, such, all available measured data are displayed.

[Data time window]

If a value is entered and confirmed by `ENTER` and **[exact]** is activated, measured data are shown for the given time window (with the point in time given by the time slider centered in the time window). This is useful, if measured data are sparse. If no value is given here, it defaults to `1`.

[new graph]

This button activates a new graph. This button is redundant. The same action can be activated by a right mouse click on the variable/box.
You may have up to nine graphs on in the drawing area, which will be automatically arranged.

[erase graph]

This button erases the actual graph with all its sets. The "new" last graph becomes active again. This button becomes active after pressing **[new graph]** or if two or more graphs are displayed in the drawing area. It becomes inactive after pressing **[erase all graphs]** or if only one active graph remains in the drawing area.

[erase all graphs]

This button erases all graphs with all their sets. This is useful to clear up the drawing area if you are loosing the overview.

[eval layer]

If **[eval layer]** is selected and if the result file contains information on the layer thickness (stored in the variable `layer`), the information on time changing layer thicknesses are evaluated to compose the plot of the profile. If the variable `layer` is not stored or if **[eval layer]** is not activated all layer thicknesses are assumed to be equal and constant.

[show layer value]

If **[show layer value]** is selected and if the result file contains information on the layer thickness (stored in the variable `layer`), the information on time changing layer thicknesses are evaluated and the depth (starting from top) is displayed on the layer axis instead of the layer number.

[flip x-y]

Change to a plot with exchanged concentration and layer axes.

[same x-axis]

If this option is active all graphs in the drawing area get the same scaling on the x-axis. This is useful if the same variable is drawn for different boxes.

[scale to time maximum]

If this option is active the x-axis maximum is set to the maximum over the whole time interval.

[Barchart]

The data will be displayed as barcharts instead of lines.

[smooth]

If **[smooth]** is active, the line segments are rendered with a set of Bezier splines otherwise they will be displayed as a polygon.

[direct apply]

If **[direct apply]** is active, all clicks in the box selector or the variable selector, as well, as all actions on the different sliders will be immediately evaluated.
If **[direct apply]** is not active, the actual settings (also creation of new sets or graphs) will be evaluated when the button **[Apply]** is pressed. Such, it is possible to prepare more complex displays in a kind of 'offline' mode and get the results displayed in one evaluation.

[Analysis]

See chapter 12.

[Command]

See chapter 13

[Set Label]

Additional text can be added to the plot in different fontsizes. The text must be given in the text field. After confirming this by `ENTER` the text appears in the drawing area. It can be moved by pressing the left mouse button on it and moving the mouse holding the button pressed. After releasing the mouse button the title jumps to that position. The label can be removed by a right mouse click while holding the shift button pressed.
The label will only appear in the PostScript resp. pdf outputs!

10 Phase Tool

This tool allows the investigation of f.e. predator-prey interactions within the model results. 3D plots of variables against each other are possible.

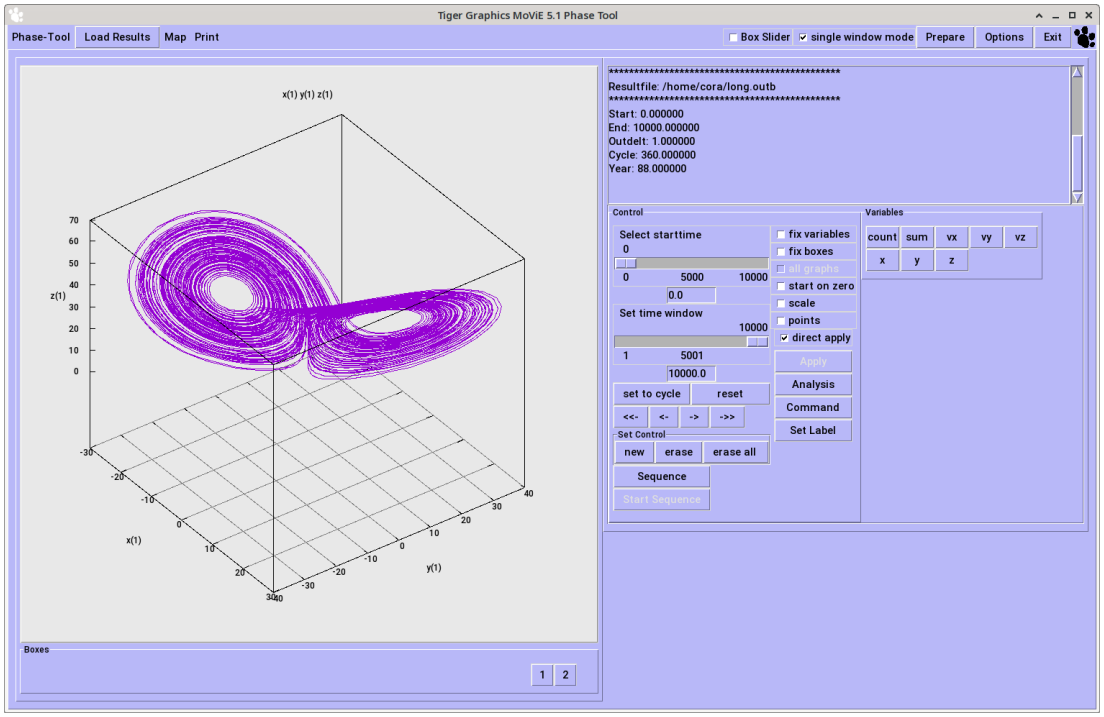

Figure 10.1: Phase tool.

Phase control panel

The control panel contains all buttons and sliders for modifying the actual plot and the plotting mode.

[Select starttime] and [Set time window]

The starting time of the plot and the period which shall be plotted can be selected. If the time window exceeds the endtime for a starttime the time window will be reduced accordingly.

[set to cycle] and [reset]

The time window is set to the value given by the variable cycle (default 360). If the variable cycle is set in the simulation parameters of the result file (see cem manual), this value is read. Reset reetsets the time window to the maximum length. **[<]** , **[>]** allows stepping cycle by cycle, **[<<]** ,**[>>]** jumps to the first resp. last cycle.

[new set]

This button activates the next plot line (set). The same action can be activated by a middle mouse click on the variable/box.
This new set remains active for all actions (f.e. from **[analysis]**) until

- another set is activated (**[new set]** or left mouse click onto a variable/box)
- it is erased (**[erase set]** or right mouse click onto the set)
- a new graph is activated (**[new graph]** or right mouse click onto a variable/box)
- the actual graph is erased **[erase graph]**
- all graphs are erased **[erase all graphs]**

You may have up to 9 sets per graph.

[erase set]

This button erases the actual (the last) set from the actual graph. After this the "new" last set of the actual graph becomes active again.

[erase all]

This button erases the all sets from the actual graph. After this the "new" last set of the actual graph becomes active again.

[fix variables]

The variable selector will become inactive and the present selection of variables will be fixed for the next actions. After selecting a new box the plot will display the fixed variables for the selected box. Clicking again on **[fix variables]** deactivates this mode and the variable selector becomes active again.
If **[new graph]** is pressed whilst **[fix variables]** is active, all information is piped from the previous graph to the new one except the box numbers which are replaced by the next selected one.

[fix boxes]

The box selector will become inactive and the present selection of boxes will be fixed for the next actions. After selecting a new variable the plot will display the selected variable for the fixed boxes. Clicking again on **[fix boxes]** deactivates this mode and the box selector becomes active again.
If **[new graph]** is pressed whilst **[fix boxes]** is active, all information is piped from the previous graph to the new one except the variable name which are replaced by the next selected one.

[act on all graphs]

This button becomes active if **[fix boxes]** or **[fix variables]** is selected.
If **[act on all graphs]** is pressed the fixes in variables or boxes will act on all displayed graphs. Pressing **[act on all graphs]** again deactivates this option. The fixes will work on the last (the actual) graph only again.

[start on zero]

The y-axis starts on zero even if there are negative data

[scale]

If active scaling of the axis is evaluated with respect to the current time window.

[direct apply]

If **[direct apply]** is active, all clicks in the box selector or the variable selector, as well, as all actions on the different sliders will be immediately evaluated.
If **[direct apply]** is not active, the actual settings (also creation of new sets or graphs) will be evaluated when the button **[Apply]** is pressed. Such, it is possible to prepare more complex displays in a kind of 'offline' mode and get the results displayed in one evaluation.

[Analysis]

See chapter 12.

[Command]

See chapter 13

[Set Label]

Additional text can be added to the plot in different fontsizes. The text must be given in the text field. After confirming this by ENTER the text appears in the drawing area. It can be moved by pressing the left mouse button on it and moving the mouse holding the button pressed. After releasing the mouse button the title jumps to that position. The label can be removed by a right mouse click while holding the shift button pressed.

The label will only appear in the PostScript resp. pdf outputs!

11 Contour Tool

This tool needs a click to **[Apply]** by default!

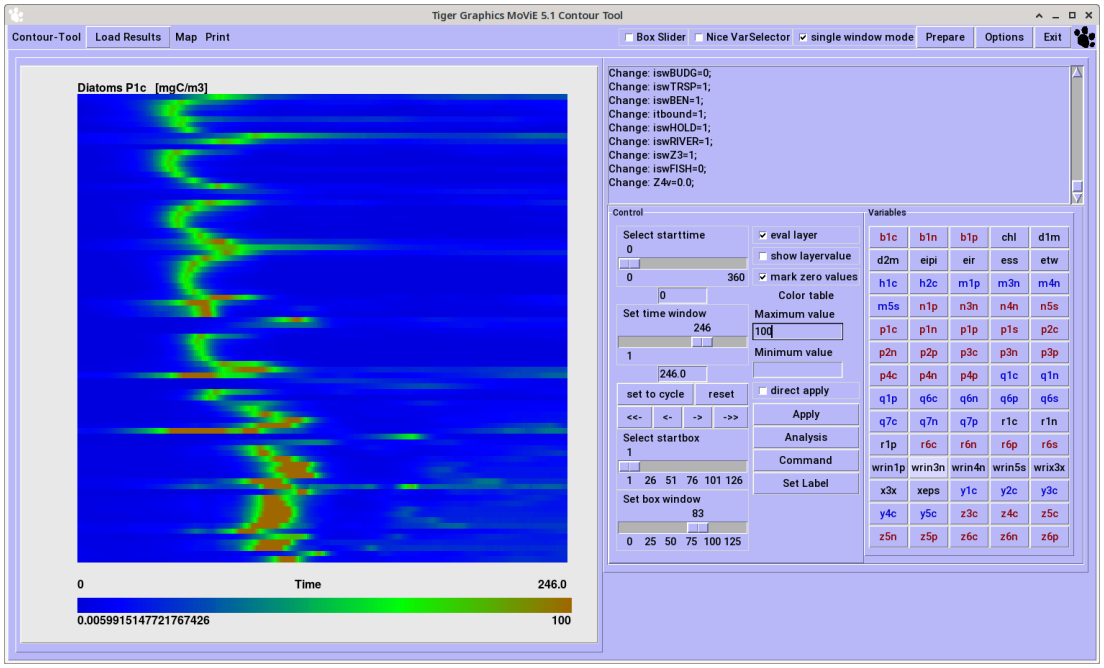

Figure 11.1: A snapshot from the Contour tool.

The Contour Tool is a more specialized tool to work with simulation results from layer oriented models. It gives an overview on the complete development of concentrations and layer thicknesses over a time window. The time information is given on the x-axis, the layer (thickness) information is given on the y-axis (starting from top) and the concentration of the displayed variable is coded by color.
To adjust the different axes to the region of interest the following controls are implemented.

Contour control panel

[Select starttime] and [Set time window]

The starting time of the plot and the period which shall be plotted can be selected. If the time window exceeds the endtime for a starttime the time window will be reduced accordingly.

[set to cycle] and [reset]

The time window is set to the value given by the variable cycle (default 360). If the variable cycle is set in the simulation parameters of the result file (see cem manual), this value is read. Reset reetsets the time window to the maximum length. **[<]** , **[>]** allows stepping cycle by cycle, **[<<]** ,**[>>]** jumps to the first resp. last cycle.

[Select startbox] and [Set box window]

The first displayed box and the numbers of box to follow can be selected.
If the time window exceeds the maximum number of boxes for startbox the box window will be reduced internally.

[eval layer]

If **[eval layer]** is selected and if the result file contains information on the layer thickness (stored in the variable `layer`), the information on time changing layer thicknesses are evaluated to compose the plot of the profile. If the variable `layer` is not stored or if **[eval layer]** is not activated all layer thicknesses are assumed to be equal and constant.

[show layer value]

If **[show layer value]** is selected and if the result file contains information on the layer thickness (stored in the variable `layer`), the information on time changing layer thicknesses are evaluated and the depth (starting from top) is displayed on the layer axis instead of the layer number.

[mark zero values]

If activated zeros are displayed in grey and not in a color of the color table.

[Color table]

Opens a menu with different color tables which can be selected.

[Set minimum] and [Set maximum]

These buttons affect the color scaling. By default the minimum and maximum values are taken over the whole stored simulation time for the selected boxes scheme (lower or upper). In the text fields the minimum and maximum values may be set by hand. All values above the maximum (below the minimum) are drawn in the "maximum" ("minimum") color.

[direct apply]

If **[direct apply]** is active, all clicks in the box selector or the variable selector, as well, as all actions on the different sliders will be immediately evaluated.
If **[direct apply]** is not active, the actual settings (also creation of new sets or graphs) will be evaluated when the button **[Apply]** is pressed. Such, it is possible to prepare more complex displays in a kind of 'offline' mode and get the results displayed in one evaluation.

[Analysis]

See chapter 12.

[Command]

See chapter 13

12 [Analysis]

This button opens a separate window which holds the functions of the analysis tool.

[+] [*] [/]

The next selected variable is added/multiplied to the previous resp. the previous is divided by the selected one.
If **[new set]** or (**[new graph]**) has been pressed before, the result will appear in the next set resp. graph.

If you like to add f.e. x(10)+x(20), press **[x]** in the variable selector, then press **[10]** in the box selector, press now **[+]** and then **[20]** in the box selector.

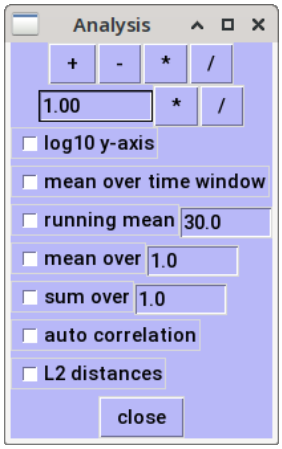

Figure 12.1: Analysis control panel

It is not possible to add f.e. x(10)+y(20).

[.....] [*] [/]

A factor may be specified and confirmed by **[return]** . If then **[*]** (**[/]**) is pressed the next variable will be multiplied (divided) by this factor.

Attention: Do not forget to confirm the factor by ENTER.
Remark: The operations may be combined: f.e. **[+]** and **[.....]** **[*]** may be pressed immediately after another; then the next selected variable is first multiplied and then added to the previous one.
Division by zero causes a wave error which doesn't affect the session. The numbers divided by zero won't change. This may result in misleading plots!

[mean over time window]

The mean over the adjusted time window (**[Select starttime]** and **[Set time window]**) is displayed as one barchart for each selected variable. All other features are still active in this mode. The **[mean over time window]** mode is active as long as the radio button is marked red.
When used in the text tool (see 7) this function also calculates some more statistical values (median, maximun, minimun, standard deviation, etc.).

[running mean]

The running mean over the selected time window is displayed for each selected variable. The length of the time window can be set in the text window on the right hand side of the label. It must be confirmed by ENTER.
All other features are still active in this mode. The **[running mean]** mode is active as long as the radio button is marked red.

[mean over]

The mean over the time window of the given length is displayed for each selected variable. The length of the time window can be set in the text window on the right hand side of the label. It must be confirmed by ENTER.
All other features are still active in this mode. The **[mean over]** mode is active as long as the radio button is marked red.

[sum over]

The sum over the time window of the given length is displayed for each selected variable. The length of the time window can be set in the text window on the right hand side of the label. It must be confirmed by ENTER.
All other features are still active in this mode. The **[mean over]** mode is active as long as the radio button is marked red.

[L2-distances]

Remark: This feature is only active in the environment of the compare tool (5.1) and if result files with spectrum data are loaded (see 14).
A new selection window appears, to select the L2-distances to be calculated. The selected L2-distances are calculated and displayed in the output windows of the L2-distance tool.

[close]

Closes the analysis control window.

13 [Command]

This button opens a window that enables the user to produce outputs with a maximum of nine predefined commands (see Figure 13.1).

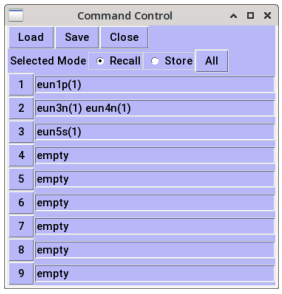

Figure 13.1: Command control panel.

Command sequences for up to nine graphs with each up to twenty variables to be displayed can be stored in files and read from files, such the user is able to setup often used sequences for different purposes without starting from scratch with every **MoViE** session.
In addition to the very simple formulas which can be produced with the Analysis tool (see section 12), the command tool offers the possibility of editing formulas in the command text fields. Those formulas are then interpreted by **MoViE** , such, the complete range of mathematical expressions (including brackets) is allowed.

Note: These formulas must not contain any white space, since this is the set separator within one graph!

Note: The user has to take care for the correctness and the syntax of those formulas. In case of errors **MoViE** will not crash but will display an error message in the Info Window, which might not be very helpful.

[load commands] and [save commands]

A file selector box is opened from where the command lines can be read or stored from/into an external file.

Files holding those commands have the extension `.mcf` (**MoViE** -Command-File).

By default `.mcf` files are either stored in the actual working directory from where the **MoViE** session was started or if existing in a directory `mcf` in the actual working directory. The structure of the `.mcf` is given in 16.6.

[Close]

Closes the command tool window. The actual commands will be held in the text lines until the **MoViE** session is finished.

[Recall] /[Store]

This button changes the mode of work of the command tool.

If the button shows **[Recall]** the numbered buttons below and the **[All]** button right of this are recalling the command sequence(s) shown in the text lines of the command window.

If the button shows **[Store]** the command sequence of the actual graph can be stored into a text line by selecting one of the numbered buttons **[1]** -**[9]** .

If **[All]** is pressed command lines corresponding to all visible graphs are stored into the adequate number of text lines starting with **[1]** . This may overwrite existing commands!

Command lines can be saved into a file by **[save commands]** and read from files by **[load commands]**

[Store] does not stay active after one storing action, but returns automatically into **[Recall]** mode!!

Commands for up to 20 sets per graph can be recalled/stored.
One text line resp. one single button **[1]** -**[9]** represents one graph.

[All]

This button acts differently depending on the selected mode shown in the **[Recall]** /**[Store]** button left of it.

In the **[Recall]** mode this button displays graphs produced by all non "empty" command lines shown in the text lines below.

In the **[Store]** mode this button stores the sequences that produced all visible graphs into the text lines below, from where those sequences can be recalled or stored into a file.

[1] -**[9]**

These buttons act differently depending on the selected mode shown in the **[Recall]** /**[Store]** button above them.

In the **[Recall]** mode these buttons display one graph produced by the command line shown in the text line right of it. Normally the last (the actual) graph is replaced by a graph built by the sequence of commands shown in the text line.

If **[new graph]** has been pressed before, a new graph is produced with those commands.

In the **[Store]** mode these buttons store the sequence of the actual graph into the text line at the right, from where the sequence can be recalled.

Advanced user may also edit the command lines. In this case only the limited capability concerning the calculus of **MoViE** is allowed.
Remark: In some tools only the first set resp. graph of the commands will be evaluated!

14 Prepare

Prepare provides auxiliary tools for transforming result and data files.

info window

All program output is shown here if the program is not started in background.

start in background

If activated all programs are started in background.

[result to fourier]

Transforms a whole `*.outc` file into spectral values and stores them in a `_pk.outc` file. The transformation method is described in Krug (1991). The pk-files are read by every tool and will be synthesized automatically. They built the base for the functionality of the L2-Distance functions. The compare tool and the text tool provide the functionality of showing the spectral values themselves.
The program doing the transformation calculates a transformation matrix, that is only dependent on the number of data samples. Since the calculation of this matrix may take a quite long time (up to some hours) it is stored in a directory

`.MOVIE-matrices` to be re-used for transforming result files with the same number of samples.

[data to fourier]

Transforms a whole `*.symb` file into spectral values and stores them in a `_pk.outc` file. The transformation method is described in Krug (1991). The pk-files are read by every tool and will be synthesized automatically. With this transformation results and field data become mathematically comparerable. This is the base for using the functionality of the L2-Distance functions on results and data. The compare tool and the text tool provide the functionality of showing the spectral values themselves. After selection of the `*.symb` file **MoViE** asks for some basic data for transformation. These values has to be adapted to the intended use of the created `_pk.outc` file:

cycle : This value defines the fundamental wavelength which will be used for the synthesis of the analyzed data sets.

outdelt : This value defines the resolution of the synthesis. The default is `0.1`.

epsi : This value controls the smoothness of the analysis of non-equidistant data sets. The default value `-1` gives control to the transformation routine to evaluate this value from the statistical variance of the data. This is a good choice in most cases. For manual adjustments of this parameter see Krug (1991).

number of harmonics : This parameter determines how many harmonics shall be stored in the `_pk.outc` file. The default value `-1` gives control to the transformation routine to evaluate this value from the number of samples according to the sampling theorem. This is a good choice in most cases.

All values can be put into the according text fields and must be confirmed by `ENTER`. The data in the `*.symb` file may be sorted in any order over time.

[Cancel]

[Cancel] stops the selection and the process.

[OK]

[OK] sends the a.m. values to the transformation routine and starts the transformation.

[result to csv] and [data to csv]

Converts `*.outc` resp. `*.symb` files into `.csv` files (comma separated values). This format can be read by most of all spreadsheet programs. The files are named similar to the source files but with the extension `.csv`.
Remark: If you are using the German version of a spread sheet program you must take care that it accepts dots instead of commas as decimal delimiter.

[csv to result]

Converts `.csv` files into `*.outc` files. The following convention for the must be fulfilled:

1. There must be one head line containing variable names not longer than twelve characters.
2. The variable names in the head line may contain a box number in parenthesis (e.g.`N1p(12)`). If no box number is contained, the box number `1` will be applied automatically.
3. The first column may contain time information. This time information must consist of equidistant samples. The according variable name in the head line must be `time` or `t`.
4. All variables must have the same number of samples.

[csv to data]

Converts `.csv` files into `*_csv.symb` files. The resulting files have the structure of `*.symb` files.

[result to data]

Converts `.outc` files into `*.symb` files.

[data to zero.outc]

Converts a `.symb` files into a special `*_zero.outc` file. The resulting file contains all variables defined in the `.symb`. The values are all set to zero. This file is useful if only the data should be visualized.

[data to outc]

Converts a `.symb` files into a `*_symb.outc` file. The resulting file contains all variables defined in the `.symb` file. The values are linearily interpolated. A popup window asks for startime, endtime, outdelt and cycle. If the intervall of the resultfile exceeds the data, the values for the missing time stamps at the beginning and at the end are set to -9999.0.
The data can be sorted in any order over time. If the data originate from several years, the product of the number of years with the value of `cycle` must be specified for the `endtime` value.

Distance Files

Calculates the pointwise distance of two files. The distances of `file1.outc` and `file2.outc` are stored in a new file named `file1_file2_m1.outc`.
Remark: The two file must be stored with the same `outdelt`!

15 Options

Options serves to set some defaults for easier work with **MoViE**. It reads information from one central **MoViE** resource file (`.tkmoviedefaults`), which is located in the user's home directory or from a local file `.tkmoviedefaults` in a model directory depnding on settings in).

15.1 File selections

General functions:
[Search] opens a file selection dialog, where files can be selected.
[Empty] removes an entry from the setting under regard.
Entries in the text fields can be made by keyboard, as well. They have to be comfirmed by pressing `ENTER`.
The following pre-selections are possible:

[Default Data Directory]

Here the path to the top level directory can be set, where measured data are stored.

Figure 15.1: Options control window

[Default data file 1] to [Default data file 5]

Here a set of data files with measured data (compare 16.5) can be preselected for model evaluation.
These files don't need to be in the path as set in **[Default Data Directory]** , but the **[Search]** starts at point in the directory tree.
The here selected files are available for the Compare tool (see 5.1), the text tool (see 7) the ProfCom tool (see 9) without reloading them when starting **MoViE** for new.

15.2 Other settings

Canvas Size

The appearance of **MoViE** can be adjusted to the available resolution of the screen. Due to the fact, that plot window and the text window (for numerical output) need

the most space, the appearance is controlled by the **[canvas size]** .
The canvas size can be optimized in eight steps for screen resolutions from `400*400` to `1600*1600` pixels.

Remember last tool

If this question is answered in the affirmative, the next time movie is called, it will be started with the last tool used. Otherwise, movie always starts with the overview tool.

[Case sensitive result files]

When selecting case sensitive result files, **MoViE** will distinguish variable according to the case of characters. E.g. `N1p` will be handled as a different variable than `n1p` or `n1P`.
It may be useful to have an automatic transformation of variable name to lower case by **MoViE** .
Remark: This will not effect any data in the result files or data files.

[X-axis representation]

When selected **[Days]** the time axis will get ticks and tick labels with months and years. It is assumed, that a year consists of twelve months with each thirty days.
The normal setting is **[numbers]**

[GUI Color Scheme]

The background color of the GUI can be changed here. Four schemes are available.

[Plot Color Scheme]

The colors of the sets in a graph can be changed here. Five schemes are available.

[Where to store preferences]

The preferences of set in this option window can either be stored locally in a model directory or globally in teh usre's home directory. In the last case the these options are valid for all models.

[Button/Label Font size]

Font size of the control panel.

[Selector Button Font size]

Font size for the variable selector and the box selector.

[Plot font size]

Here three font size for the texts in graphical outputs can be defined. There are settings for three different display variants:

normal The here selected font is used for the legends and the axes when displaying one or two graphs in the plotting window of **MoViE** .

small The here selected font is used for the legends and the axes when displaying three or four graphs in the plotting window of **MoViE** .

tiny The here selected font is used for the legends and the axes when displaying five to nine graphs in the plotting window of **MoViE** .

Remark: The Plot font size has an effect on Postscript outputs but not on **xmGrace** outputs.

15.3 Tool control

[Reset to defaults]

Resets the options to default values.

[Cancel]

Clicking **[Cancel]** leave this tool without applying any changes to the `.tkmoviedefaults`

[Apply&Exit]

Clicking **[Apply&Exit]** wrtites the settings to `.tkmoviedefaults`, applies them and closes the window.

16 Data structures

16.1 xxx.outa

The simplest data structure that **MoViE** can read as a result file are comma separated ASCII `.outa` files. File names must not contain any special characters. A valid `.outa` file must contain seven header lines:

```
Stored Variables:<number>
Start Time:<number>
End Time:<value>
Outdelt:<value>
Year:<value>
Cycle:<value>
time,<var1>,<var2>
```

The number of stored variables must match the number specified in line 1. Start time and Endtime can be given as integer or floating points. Outdelt must be given as integer ot floating point and must match the difference between to time steps. Year and Cycle are needed as lines but can be empty. Cycle is used for displaying special x-grid lines, yera for comparison with data. Variable names must not contain any special characters and must start with a letter .The variable names may have a box specification (index), f.e. `B1c(1)`. A mix of indexed and un-indexed variables is not allowed. The data must start in line eight starting with a time step followed by the values of the variables at that time, all separated by commas.
Valid result files are f.e.

```
Stored Variables:6
Start Time:1.000000
End Time:7.000000
Outdelt:1.000000
Year:2007.000000
Cycle:365.000000
time,ETW(1),PAR(1),Np(1),Nn(1),Ns(1),CHL(1)
1.00,4.498312,6.018057,0.700000,35.000000,28.000000,2.600000
2.00,6.924421,5.146552,0.724273,35.388363,28.157314,2.514765
3.00,6.826476,3.564123,0.758683,35.938923,28.393526,2.392004
4.00,6.607920,3.089755,0.794103,36.505642,28.650551,2.220717
5.00,7.044028,2.684833,0.829489,37.071823,28.915953,2.050570
6.00,7.179232,2.465950,0.866498,37.663967,29.195692,1.878965
7.00,7.162212,3.079893,0.903312,38.252998,29.476221,1.717669
```

```
Stored Variables:2
Start Time:0.000000
End Time:1.000000
Outdelt:0.100000
```

```
Year:0.000000
Cycle:0.000000
time,R,B
  0.00,20.000000,200.000000
  0.10,20.060106,200.039764
  0.20,20.120424,200.079056
  0.30,20.180956,200.117874
  0.40,20.241701,200.156204
  0.50,20.302660,200.194061
  0.60,20.363832,200.231430
  0.70,20.425219,200.268326
  0.80,20.486822,200.304733
  0.90,20.548639,200.340637
  1.00,20.610674,200.376068
```

In the case of large data sets, there may be a delay when reading in. To speed up working with **MoViE**, `.outa` files are initially converted to `.outc` files (16.2) and stored in the directory under the name `<resultname>_outa.outc`.

16.2 xxx.outc

Files with an extension `xxx.outc` (**CEMoS** default) are holding model outputs of **CEMoS** .
Variable names and the path to the model directory in `xxx.outc` files may consist of 80 characters, each.

The `.outc` file starts with structural information given in different formats:

(1) total number of stored variables (`nvars` automatically derived by **CEMoS**)

(2) start time of the simulation (`start` read from `cemos.par`, normally 0 for January 1 of year (see (6))

(3) end time of the simulation (`endtim` read from `cemos.par`)

(4) maximum timestep of the simulation (`maxdelt` read from `cemos.par`)

(5) simulation time between two outputs (`outdelt` read from `cemos.par`)

(6) year, where the simulation starts (`year` read from `cemos.par`, stored is max(0,year))

(7) length of a model's year (`cycle` read from `cemos.par`, normally 360 days representing 12 months of 30 days.)

(1) is stored as unsigned integer, (2) - (7) are stored as floats in binary format.

(8) relative path to the main model's directory, stored as string of 80 ASCII characters read from `cemos.par`)

(9) This header is followed by a set of `nvars` strings of each 80 ASCII characters. These strings are containing the name of a variable and the box number for which it is stored (f.e. `'p1c(137)'` indicates p1c of box 137 being stored.)

(10) After this block of strings (which may be some ten kilobytes long) the numerical output of the simulation is stored as follows:
One float is stored in binary format for each variable-box-combination appearing in the string block (9). One block is stored for simulation times between `start` (2) and `endtim` (3) that are multiples of `outdelt` (5). This block may have a length of several megabytes.

16.3 xxx_pk.outc

Files with an extension _pk.outc are holding spectral values from Fourier analyzed result files. (see 14)

The _pk.outc file starts with structural information given in different formats:

(1) total number of stored variables (nvars automatically derived by **CEMoS**)

(2) start time of the stored data

(3) end time of the stored data

(4) number of base vectors (one base vector means one data pair for each harmonic) as calculated by the Fourier transformation.

(5) outdelt for synthesizing the data.

(6) number of cycles the Fourier transformation was calculated for.

(7) length of the basic cycle (length of the first harmonic)

(1) is stored as unsigned integer, (2) - (7) are stored as floats in binary format.

(8) the word fourier as a string of 80 characters to identify that this file contains spectrum data (the word fourier is filled with spaces to make up a string of 80 resp. twelve characters).

(9) This header is followed by a set of nvars strings of each 80 ASCII characters. These strings are containing the name of a variable and the box number for which it is stored (f.e. 'p1c(137)' indicates p1c of box 137 being stored.)

(10) After this block of strings (which may be some ten kilobytes long) the spectrum data are stored as follows:
One float is stored in binary format for each variable-box-combination appearing in the string block (9).
One block is stored for each base vector from 0(!) to number of base vectors (4).
This block may have a length of several megabytes.

16.4 xxx.nc

MoViE also reads **NetCDF** files of the format described in the following. A complete specification of the **NetCDF** interface in the used version **3.5.1** is given in (NetCDF, 1997). Here, the variables and their types as used by **CEMoS** and read by **MoViE** are given. For the data formats of NC_CHAR, NC_INT and NC_FLOAT see (NetCDF, 1997).

[global attributes]
One global attribute is set by **CEMoS** : NC_CHAR originator = "created by TIGERGRAPHICS CEMoS"

[dimensions]
The dimension of time information appears as follows time_dim = UNLIMITED, this means, that the simulation output is written sequentially, although the number of store events is determined by starttime, endtime and outdelt.

[variables]
The following variables are describing the the structure of the .nc file (similar to 16.2):

[NC_INT varnumber] total number of stored variables

[NC_FLOAT starttime] start time of the simulation

[NC_FLOAT endtime] end time of the simulation

[NC_FLOAT maxdelt] maximum timestep of the simulation

[NC_FLOAT outdelt] simulation time between two outputs

[NC_FLOAT year] year, where the simulation starts

[NC_FLOAT cycle] length of a model's "year"

[NC_CHAR model_dir(80)] relative path to the main model's directory, stored as string of 80 ASCII characters.

[NC_FLOAT time(time_dim)] simulation time do reference from the position in the file to the simulation time.

[NC_FLOAT proceed] this variable holds the actual simulation time of the simulation run, when reading results from a still running or from an interrupted simulation. When a simulation is finished it holds the same value as endtime.

All variables are expected as a combination of `name(box)` with the same dimensions as `time`, where the **NetCDF** interface keeps track of the actual stored amount of time steps. Such, if the variable `X` should be stored for the boxes 1 and 2, the following shall appear in the `.nc` file:

[NC_FLOAT X(1)(time_dim)] and

[NC_FLOAT X(2)(time_dim)]

The setting `time_dim` ensures the coupling of the variables to the simulation time stored in `time`.

The approach of storing each variable-box combination as **one NetCDF** variable keeps the **CEMoS** capabilities of selecting fit to interest sets of store variables instead of storing a variable for **all** boxes.

NetCDF files are containing the same amount of information as the `.outc` files. Information about the content of an `.nc` file can be shown by typing the command `ncdump <filename>` in a terminal window. Information about the usage of that program will be displayed by simply typing `ncdump`.

Note: The `.nc` file format is mainly used to port **CEMoS** results to other environments. The special functions (Fourier transformation, converting to `.csv`, etc.) of the Prepare Tool (see section 14) are actually working with the `.outc` resp. `.outb` format, only.

Note: The access to variables in **NetCDF** files works case sensitive on their names, such, when working with a mix of `.nc` and `.outc` files, it will be neccesary to switch to case sensitive handling of results (see 15.2). (This automatically happens, when an `.nc` file is loaded on start of **MoViE**).

16.5 xxx.symb

The files with the `.symb` extension are holding validation data for model variables (see chapter 5).
Each set of validation data consists of minimum two lines with an identical number of columns. The lines contain the time information and the value. In addition, lines for standard deviation, validity range of the value, time of day are possible.

1st column: from the beginning to the first '/' contains the name of the variable and the boxnumber (in brackets).

2nd column: between first '/' and second '/' denotes the meaning of the row:

/tim/ time information as numeric value.

/ymd/ Time information as days as alternative to /tim/. The format is YYMMDD (f.e 890815 means August 15, 1989).

/tid/ Additional row to /ymd/. Indicates the time of day. The format is hhmm (f.e. 1430 indicates 2:30 pm).

/val/ indicates the value(s) of the measurement(s) to follow.

/std/ indicates the standard deviations of the values.

/int/ /inb/ alternative to /std/ indicate non-symmetrical quantiles instead of the symmetrical standard deviation, where /int/ denotes the value above /val/ and /inb/ denotes the value below /val/ to be taken for the error bar in the plot.

/per/ indicates the length of the interval of validity for /val/ given at /ymd/.

16.6 xxx.mcf

`.mcf` files contain predefined sets of variables (see 13.1). Every line within a file corresponds to a graph. Within a line the sets are separated by whitespaces. Up to nine lines are possible. Empty lines should be marked by `empty`.
Example:

```
n1p(1) n1p(2) n1p(3)
n4n(1) n4n(2) n4n(3)
n4n(1)/n3n(1) n4n(2)/n3n(2) n4n(3)/n3n(3)
p1c(1)/100.0
empty
empty
empty
empty
empty
```

16.7 name.info

In this file the information is stored about the variables that may occur in the user's model output. The file contains pure ASCII data. Those data shall be organized as follows:

1st column: (Short) name of the variable appearing in the model's output.

2nd column: Description of the variable's meaning.

3rd column: Unit of the variable.

4th column: Identifier for type, e.g. benthic or pelagic or none.

5th column: Identifier for the index ("box" number, e.g. p1c(134)) to identify, whether this number should be understood as a box number EulerBox, as a voxel number Voxel or as a layer number Layer.

The columns must be separated by a colon ':', such, the colon is not allowed in a name of a variable, a description or a unit.

For example:
N1p:Phosphate N1p:mmol/m3:pelagic:Voxel
results in the following head line for e.g. *N1p(13)* in **MoViE** canvases:
Phosphate N1p Voxel 13 [mmol/m3]
and
A2c:Benthic Non-Diatoms A2c:mgC/m2:benthic:EulerBox
results in the following head line for e.g. *A2c(25)* in **MoViE** canvases:
Benthic Non-Diatoms A2c EulerBox 25 [mgC/m2]

The "normal" head line (in absence of name.info) will simply be *N1p(13)* and *A2c(25)*.
One default name.info comes with the **MoViE** package. It is located in TKMOVIEHOME/info.
If a name.info is present in a model directory, this local one will be used instead of the default one.

16.8 nicevar.info

An alternative variable selector (nice variable selector) which contains other names instead of variable names can be constructed. For this a file named nicevar.info must be available in the directory where **MoViE** has been.
These file must have the following structure:

First line: number of columns of buttons in the selector

Further lines: variables or seperator (=).
A line for a variable must have the following structure:
<variable name>:<new name>:<color>

The color might be any **Tcl/TK** color (see: man colors).

```
Example:
2
irr:Irradiance:navyblue
temp:Temperature:navyblue
=
chl:Chlorophyll:seagreen
zoo:Zooplankton:seagreen
```

The normal variable selector and the nice variable selector can be toggled by clicking the radio button in the top menu.

16.9 .tkmoviedefaults

The file is located in the home directory an in addition in the actual model directory. It contains the settings of the custom tool (15). It is written if the settings of the custom tool are applied. If the file exists in the actual model directory this one will be evaluated otherwise the file from the home directory. If no `.tkmoviedefaults` is available the **MoViE** defaults will be set.

16.10 .tkmoviemcf

This file is located in the actual model directory. If existing, it contains the last commands of the command window. These settings will be restored if the command window is opened again.

16.11 .tkmovieset

This file should only be created by advanced **MoViE** users. This file must be located in the home directory. It may contain default settings for several global **MoViE** variables. The file contains **Tcl/TK** code which sets the variable to the specified value. The file must not contain any other lines! It is allowed to append a **Tcl/TK** comment after a `;`.

Example of `.tkmovieset`

```
set prettyaxis 0
set sameaxis 1;# sets all y-axes to the same value by default
```

References

[Hamberg 1996] Hamberg, F.: *CEMoS, eine Programmierumgebung zur Simulation komplexer Modelle*, Fachbereich Mathematik, Carl von Ossietzky Universität Oldenburg, Diplomarbeit, 1996

[Hamberg und Kohlmeier 2023] Hamberg, F. ; Kohlmeier, C: *CEMoS 5.0*. 2018. – URL http://www.staff.uni-oldenburg.de/cora.kohlmeier/software/cemoscemtk.pdf.

[Krug 1991] Krug, P.: The conditional-expectation as estimator of normally distributed random-variables with values in infinitely dimensional banach-spaces. In: *J. Multivar. Anal.* 38 (1991), S. 1–14

[NetCDF 1997] Rew, R. ; Davis, G. ; Emmerson, S. ; Davies, H.: *NetCDF User's Guide for C Version 3*. Unidata Program Center: , 1997