

The non-hydrostatic mesoscale model

MEMO

Version 6

Input Interface Reference

Laboratory of Heat Transfer
and Environmental Engineering
Faculty of Engineering
University of Thessaloniki

October 2005

Introduction

In order to set up a MEMO run, various files containing input data and configuration parameters should be prepared and placed in the appropriate directories. The following example assumes a nested run for a city named “CITY”:

```
./                                ! Base directory                [DIRECTORY]
    config.dat                    ! Configuration file            [INPUT, see
section 4]
    info.cas                      ! Case file                      [INPUT, see section 3]
    path.fil                     ! Path-description file         [INPUT, see section 6]
    soundnew05.1.met             ! Meteorological (soundings) file [INPUT, see section 5 ]
    topomemo_4.dat              ! Topography & landuse file, coarse [INPUT, see section 2]
    topomemonest_1.dat         ! Topography & landuse file, fine [INPUT, see section 2]
    prep6 [MEMOPREP.exe] ! Preprocessor executable file [EXECUTABLE, sect. 1]
    memo6 [MEMOMAIN.exe] ! Main MEMO executable [EXECUTABLE, sect. 1]
    plot6 [MEMOPLLOT.exe] ! Postprocessor executable file [EXECUTABLE, sect.
11]
    .m_execute                  ! Helper script                 [SCRIPT, see
section 7]
    runcase                    ! Helper script                 [SCRIPT, see section 7]
    ./OUTfiles                 ! Output files directory       [DIRECTORY]
        CITY2002_may08.1      ! Main output file, coarse [OUTPUT, see section 8]
        CITY2002_may08.2      ! Main output file, fine [OUTPUT, see section 8]
        recover.in            ! Recovery output from the preprocessor [OUTPUT,
sect. 9]
    recover.out                ! Recovery output from the main run [OUTPUT, sect.
9]
    test.end                   ! Empty file (used only as an indicator) [OUTPUT]
```

In order to avoid unnecessary and error-prone duplication, as well as waste of disk space, files common to different runs/cases should be replaced by symbolic links pointing to a common subdirectory where all the relevant files should reside. For example, different cases using the same configuration and covering the same area should include links for the Configuration and Topography files pointing to a common set of files.

Examples of Configuration, Case, Path-description, Meteorology and Topography files, as well as the shell scripts used for invoking the code, are given in subsequent sections.

1. General information about Input Data

At the beginning of each MEMO run, the preprocessor executable (`prep6` or `MEMOPREP.exe`) is called by the control script (see section 7). The preprocessor reads configuration and initialisation data from all the input files, as described below, then creates a “recovery snapshot” containing the values of state and output model variables, which is stored in “`recover.in`”.

Next, the main MEMO executable (`memo6` or `MEMOMAIN.exe`) is called. It has two inputs:

1. The initialisation “state” as provided by the `recover.in` file, generated by the preprocessor
2. The time-dependent boundary conditions, periodically generated from the soundings

(meteorology) file.

During the execution of the main executable, three time indicators are displayed at each timestep, all in seconds: the first two indicate "elapsed model time" for the two nests, coarse and fine, respectively. The third one is the CPU time from the beginning of the simulation.

Fehler! Textmarke nicht definiert.

1.1 Topography and Surface Type

The Topography file contains orography height and surface type for each grid location (i, j) with $i = 2, \dots, IM+1$; $j = 2, \dots, JM+1$. IM and JM are the numbers of grid points in x- and y-direction. The x-axis (index i) must be directed to the western direction and the y-axis (index j) to the northern direction. The grid point (2,2) is in the SW-corner of the computational domain.

The following surface types are distinguished and must be stored as percentage in the array LU(I,J,L).

L	designation
1	water
2	arid land
3	few vegetation
4	farmland
5	forest
6	suburban area
7	urban area

Only percentages for surface types 1 to 6 have to be stored. Type 7 is the difference between 100% and the sum of types 1 - 6 (section 2). If the percentage of a surface type is 100%, then write the number 10 and for all other surface types the number 99.

The orography height is the mean height above sea level for each grid location (i,j), in meters.

The precise input format and the meaning of the variables are given in section 2.

1.2 Meteorological Data

The prognostic model MEMO uses a set of partial differential equations in 3 spatial directions and in time. To solve these equations information about the initial state in the whole domain and about the development of all relevant quantities at the lateral boundaries is required.

a. Initial state

To generate an initial state for the prognostic model a diagnostic model [Kunz 1991] is applied using measured temperature and wind data.

Both, temperature and wind data can be provided as:

- i) surface measurements i.e. single measuring directly above the surface (not necessary)
- ii) upper air soundings i.e. soundings that consist of several (at least two) measurements at different height levels at a constant geographical location (at least one sounding

for temperature and wind velocity is necessary)

In section 5, a list of all input variables used in the Meteorological input file is given, together with an example input file.

b. Time-dependent boundary conditions

Information about quantities at the lateral boundaries can be taken into account as surface measurements and upper air soundings. Therefore, a key word and the time when boundary data is given must occur in front of a set of boundary information. The same input format is used as to provide initial state data.

1.3 Nesting Facility

In MEMO Vs. 5.0 a one-way interactive nesting scheme was implemented. With this nesting scheme a coarse grid (CG) and a fine grid (FG) simulation can be nested. During the CG simulation data is interpolated and written to a file (FT-UNIT41). A consecutive FG simulation uses this data (FT-UNIT 42) as lateral boundary values.

Compiling: When building up the FORTRAN-library for the CG simulation an INCLUDE-file has to be provided containing the maximum number of grid cells of the FG domain (IMN, JMN and KMN).

Controlling of CG run: Control of the nesting facility is performed from the FT-UNIT 5 file. If the current simulation is used as a CG simulation INESO must be set to 1. Furthermore the origin of the FG domain within the CG domain (XZERO and YZERO in [m]) must be given. The nesting factor, i. e. the relation ($XNEST = DX_{CG}/DX_{FG}$) of the grid spacing of both grids must be specified. The nesting factor XNEST should not exceed the value of 3.

The update time interval is given in the CG simulation by DTNESS. It should be chosen as small as possible (best results with $DTNESS = DT_{CG}$) but should not exceed $3 * DT_{CG}$.

Controlling of FG run: Within the control file of the FG simulation INESI can be set either to 1 or to 2. INESI=1 uses expanded radiation conditions for the nesting facility while INESI=2 applies a relaxation scheme. The latter being more efficient at inflow boundaries whereas expanded radiation conditions seem to be superior at outflow boundaries.

The FG simulation itself can also be used as CG simulation for a finer grid and so forth.

IMPORTANT: The altitude of the model top (ZT) and the minimal vertical grid spacing (HHMIN) must be identical in both simulations.

2. The Topography file

In the topography-file you define the complete calculation area. First you define the number of cells in the x-, y- and z-direction (IM, JM, KM). The next specification is the length of one cell in x- and y-direction (XDX, XDY). Then the height of the calculation area is defined and the partition of the area in this direction.

A separate topography file is needed for each nesting level.

Comment	Here you can write how you want title your project. This comment is also used for the title of the tecplot-output-file.
IM; XDX	<p>IM is the number of grid points in the x-direction. A minus in front of the IM-parameter means that the grid points in this direction are equidistant. The number of points must be dividable through 2 and 3 or through 5.</p> <p>The specification of XDX is in meters.</p>
JM; XDY	<p>JM is the number of grid points in the y-direction. A minus in front of the JM-parameter means that the grid points in this direction are equidistant. The number of points must be dividable through 2 and 3 or through 5.</p> <p>The specification of XDY is in meters.</p>
KM; ZT; HHMIN	<p>KM is the number of points in the z-direction. A minus in front of the KM-parameter means that the z-coordinates follow the topography. The influence of the topography to the horizontal layers decreases with increasing height. It is assumed that in the transformed coordination-system the lower boundary of the calculation area becomes a plane with a constant coordinate-value. This allows a simpler consideration of the boundary conditions at the lower border. The transformation is performed by the following equation:</p> <p>ZT stands for the total height of the calculation area. The specification is in meters.</p> <p>HHMIN defines the height of the layer above the ground. HHMIN is equivalent to in the picture below. Within the rule the value of can be chosen freely. The specification is in meters.</p> <p>The KM horizontal layers between the surface layer with a height of HHMIN and the top height ZT is calculated from MEMO itself. The following equation shows how MEMO calculates the odd layers:</p> <p>The picture below shows the calculation grid in the vertical direction.</p>

	Picture x.x: Calculation grid in vertical direction
KMB; DHB(KMB+2)	<p>KMB specifies the number of horizontal layers below the surface. Without a minus in front of KMB the layer thickness is variable. The option with constant thickness (with a minus in front) is not implemented yet. This is needed to simulate heat conductance into the ground.</p> <p>DHB is a specification of the depth of each ground layer. The number of specifications is KMB+2. The first value defines the deepest layer and the last value the top layer in the ground. The specification of the length is in meters.</p>
KMTOP; DHHUP	<p>KMTOP is the number of layers above the calculation area. This is needed for IRIS (the radiation model). A minus in front of the KMTOP-specifications means that the thickness of the layers are constant.</p> <p>DHHUP is the height of the additionally layers above the calculation area. The specification is in meters.</p>
LMAX	With this parameter the maximum number of land use classes is specified. MEMO can handle 7 different classes.
Format for reading of the land use classes and topography	In the following two rows, the format-specification for reading the land use classes and topography are listed.
Land use	Now the land use for each cell is listed, in percentages. The listing follows the following pattern:

	<p>Only six of seven land use class percentages are defined here. The seventh class is calculated as the remainder, in the subroutine topo.f90. Two Integer numbers define one land use class. The first land use definition is always used for water. The following definitions are user-defined. When a land use class covers 100% of the area in a cell, the corresponding definition looks like the following examples:</p> <p>For Water: 1099999999 and for a other class, e.g.: 991099999999 or, e.g.: 9999999109999</p>
Topography	The topography is read in a similar format as the land use classes. (see example below)

An example of the topography file for the “coarse” grid [topomemo_4.dat]:

```

! Comment Line 1
! Comment Line 2
! ...
!
!
!
[BEGIN]
! grid (72*4=288Km)*(72*4=288Km)
--CITY--

! IM   XDX
   -72  4000.00

! JM   XDY
   -72  4000.00

! KM   ZT   HHMIN
   -25 6000.00 20.00

! KMB  DHB(KMB+2)
5  1.00 0.52 0.25 0.09 0.03 0.01 0.01

! KMTOP  DHHUP

```

-5 800.00

! LMAX

7

! Format for landuse and topography

(72(6I2,1x))

(72(F7.2,1x))

! Landuse

500025250000	109999999999	109999999999	109999999999	109999999999	109999999999	109999999999
930000060000	870012000000	121856001200	008700120000	009300000600	000012256200	000062063100
000062063100	000093000600	001243370600	000600930000	001250370000	120068061200	250031430000
430012430000	180000810000	999999109999	999999109999	999999109999	999999109999	999999109999
003712500000	120031560000	680000310000	109999999999	109999999999	109999999999	109999999999
109999999999	109999999999	109999999999	109999999999	109999999999	109999999999	109999999999
109999999999	109999999999	109999999999	109999999999	109999999999	109999999999	109999999999
109999999999	109999999999	109999999999	109999999999	109999999999	109999999999	109999999999
109999999999	109999999999	109999999999	109999999999	109999999999	109999999999	109999999999
109999999999	109999999999	109999999999	109999999999	109999999999	109999999999	109999999999
109999999999	109999999999	109999999999	109999999999	109999999999	109999999999	109999999999
109999999999	109999999999	109999999999	109999999999	109999999999	109999999999	109999999999
109999999999	109999999999	109999999999	109999999999	109999999999	109999999999	109999999999
109999999999	109999999999	109999999999	109999999999	109999999999	109999999999	109999999999

[[[[[71 more lines follow]]]]

! Topography

93.69	0.00	0.00	0.00	0.00	0.00	23.31	2.00	433.69	891.50	814.19
694.69	394.75	187.19	242.25	200.88	62.31	7.25	7.44	10.44	285.56	442.69
107.81	126.50	331.00	628.56	334.69	43.00	0.00	0.00	0.00	0.00	0.00
0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00
0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00
0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00
0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00

[[[[[71 more lines follow]]]]

3. The Case file

Here a description for the Case file is given. The first column contains the label for each parameter, the second a short description.

This file is divided in two sections. The first one contains all the case-related settings that are common to all the nests:

Vers:	Version of the current MEMO-Model. This is only a comment and doesn't change.
GeoLocation	This is the definition of geographic location. The first number describes the value of the latitude, the second number the longitude and the third describes the meridian time zone. The time zone meridian will be calculated by the following equation: tzm (angle of the time zone in degree) = $360 / (LST - GMT)$ LST = Local Standard Time GMT = Greenwich Meridian Time
DateAndTime	Time declaration for the beginning of the simulation: yy.mm.dd.hh.mm yy = year; mm = month; dd = day; hh = hour; mm = minute
SimulationTime	Maximum of time (in seconds), which the simulation should last.
LogControl	Controls the periodic output of debugging information. The specification format is the same as for "PlotControl" (it's rarely used).
PlotControl	Parameters for a plot-output. This is an output for the plot-program Ittplot. First number: Type = Integer; 0 = no output; 1 = output Second number: Type = Real; Starting time of the plot-recording, in seconds. Third number: Type = Real; Timestep between two records in seconds.
TecControl	Parameters for a plot output. This is an output for the plot-program Tecplot. First number: Type = Integer; 0 = no output; 1 = output in ASCII on scalar points; 2 = output in binary format on scalar points; 3 = output in ASCII on grid nodes; 4 = output in binary format on grid nodes Second number: Type = Real; Start time of the plot-recording in seconds. Third number: Type = Real; Timestep between two records in seconds.
CouplingControl	"CouplingControl" controls the coupling between MEMO and MIMO, thus is irrelevant for the current version of MEMO.
RoughnessLength	Values of Roughness length for the several land use classes in meters. In MEMO you can define 7 land use classes, but the first class must be water.
SurfaceTemp	Starting temperature on the surface for each land use class, in Celsius degrees.
Albedo	Short wave-Albedo for each land use class
VolHeatCap	Volumetric Heat Capacity of each land use class in J/(m ³ K).
EvaporParm	Evaporation-Parameter of each land use class. $0 \leq \text{EvaporParm} \leq 1$

Vers:	Version of the current MEMO-Model. This is only a comment and doesn't change.
TempConduct	Heat Conductivity of each land use classes in m^2/s
SoilTemp	Soil temperature in a depth of 2 meters, in Celsius degree.
SEMANUControl	If you use the SEMANU-Model, you switch off the radiation model IRIS. With the SEMANU-Model you define time-dependent temperature gradients for each land use class. SEMANUControl defines a time interval after which the temperature gradients are changed. The specification is in hours.
dT/dt_i	This is the temperature gradient which applies for a certain time interval.

Second section of the Case-File. The following table shows settings which are different for each nest and each nest needs separate settings. So the number of setting-tables is the same as the number of nests.

Comment	Description of the nest
Timestep	<p>With this setting you define the time step of the calculation for a certain nest. For short calculation times, the time steps should be set as large as possible. The maximum value of a timestep is defined through the Courant-Friedrichs-Levi-Theorem, defining as an upper limit the Courant-Number:</p> <p>This equation refers to a one dimensional case. For a three dimensional case C_{max} will be calculated as the maximum of the one-dimensional Courant-numbers in the three spatial directions:</p>
Zero	<p>The location of the nests will be defined with this parameter.</p> <p>First number: Type = Real; This defines the position in x-direction of the origin of the nest inside the coarse grid.</p> <p>Second number: Type = Real; This defines the position in y-direction of the origin of the nest inside the coarse grid.</p> <p>Third number: Type = Real; This is the nesting factor. The nesting factor is defined as the ratio between the length of a mesh in the coarse grid to the length of a mesh in the fine grid:</p>

An example of the Case file [info.cas] is given below:

```

!
! Control file for MEMO 6 user defined
!
!-----
! 1. Section: common for all nests
!-----
!
! General
! -----
! Vers=          Version #
! GeoLocation=   Latitude, Longitude, Time zone meridian
! DateAndTime=   Simulation start time [yy.mm.dd.hh.mm]
! SimulationTime= Max simul. time
! LogControl=    Logfile (Switch, t_start, i_increment)
! PlotControl=   Plotfile (Switch, t_start, i_increment)
! ParticleControl= Part.file (Switch, t_start, i_increment)
! Vers=          6.0
! GeoLocation=   37.96667 23.71667 30.
! DateAndTime=   97.10.08.00.00
! SimulationTime= 345600.
! LogControl=    6 10800. 10800.
! PlotControl=   1 0. 3600.
! CouplingControl= 0 7200. 7200.
!
! Landuse Data
! -----
! RoughnessLength= Roughness length
! SurfaceTemp=     Init. Surf. Temp.
! Albedo=          Albedo
! VolHeatCap=     volumetric heat cap.
! EvaporParm=     Evapor. parm.
! TempConduct=    Temp. conductivity
! SoilTemp=       Temp. in the soil, at surface height
! RoughnessLength= 0.001 0.01 0.02 0.02 0.12 0.5 0.8
! SurfaceTemp=     15.0 15.0 15.0 15.0 15.0 15.0 15.0
! Albedo=          0.0 0.22 0.22 0.22 0.10 0.23 0.20
! VolHeatCap=     4.18E6 2.68E6 2.68E6 2.86E6 1.17E6 2.20E6 2.34E6
! EvaporParm=     1.0 0.01 0.05 0.15 0.20 0.10 0.05
! TempConduct=    0. 1.0E-6 1.0E-6 0.7E-6 0.8E-6 1.3E-6 2.0E-6
! SoilTemp=       15.0 1438.
!
! Data for diagnostic surface temp. model SEMANU
! -----
! Update time interval, # of update seqs.
! SEMANUControl= 0.1
! dT/dt for each Landuse class
! dT/dt_i=        0.0 6.0 5.0 3.0 2.0 4.0 5.0
! dT/dt_i=        0.0 0.0 0.0 0.0 0.0 0.0 0.0
! dT/dt_i=        0.0 -3.0 -2.5 -1.5 -1.0 -2.0 -2.5
!-----
! 2. Section: individual for each nest

```

```
!-----  
!  
! First Nest  
! -----  
! Comment=      Comment card  
! TimeStep=     Time step, variable (1) or not (0)  
! Zero=         FG origine within CG domain, Nest.factor  
! Comment=      ---CITY---  
! TimeStep=     10.  
! Zero=         0. 0. 0.  
  
!  
! Second Nest  
! -----  
! Comment=      Comment card  
! TimeStep=     Time step, variable (1) or not (0)  
! Zero=         FG origine within CG domain, Nest.factor  
! Comment=      CITY area simulation  
! TimeStep=     5.  
! Zero=         110000. 110000. 0.
```

4. The Configuration file

In the configuration file you define the type of boundary conditions, physical parameters, parameters for the turbulence-model and you can choose between different turbulence-models. Further you can choose different calculation methods.

In Unix, starting a file name with a dot (e.g. “.config.dat”) flags the file as "hidden" (the same as specifying the "hidden" flag in DOS).

INHYD	<p>This is a switch to decide if the calculation is running with the non-hydrostatic-pressure (changes of the pressure caused by temperature-gradients) or not.</p> <p>Yes = 1; No = 0 (Calculation only with the hydrostatic-pressure)</p> <p>For explanation :</p> <p>In MEMO the pressure is divided in three parts. One part is the pressure of the large-scale weather situation, the next is the pressure caused by the weight of the air (hydrostatic pressure) and the last one is caused by temperature gradients (non-hydrostatic-pressure). If the characteristic scale of the phenomenon is longer than 10 km, you don't need the non-hydrostatic-pressure for the calculation. This splitting effected a better behaviour in the convergence to solve the elliptical differential equation for the non-hydrostatic-pressure.</p>
INHYDNL	<p>Selection of the method for the calculation of the hydrostatic-pressure:</p> <p>=1, with subroutine HYD3NL; =2, with subroutine HYD4NL; =3, with subroutine HYD3</p>
ITURB	<p>Possible choices for the turbulence model:</p> <p><0, TTT; =1, Prandtl (analytical calculation method, 0-equation-model); =2, Schuhmacher (analytical calculation method, 0-equation-model);</p> <p>=3, k-Model (1-equation-model, for MEMO the best model); =4, k,e-Model (2-equation-model)</p> <p>·0-equation-models describe the exchange-coefficient k_m and the turbulent Prandtl-Number Pr_t with algebraic relationships as a function of the deformation D, the Richardson-Number Ri_g and of the mixing length l, which is provided to the model.</p> <p>·1-equation-models solve a transportation-equation for the turbulent energy E. The exchange-coefficient is calculated with the following Ansatz:</p> <p>The mixing length l and the turbulent Prandtl-Number Pr_t are given by a algebraic equation.</p> <p>·2-equation-models solve additionally to the transportation-equation for the turbulent-energy E a further transportation-equation, e.g. for the the dissipation e or for the mixing-length l. The algebraic specification of a mixing-length is not necessary yet.</p>
IADV	<p>Selection of advection-scheme</p>

	=1, calculation with TVD-scheme; =2, calculation with FCT-scheme
IAUTO	Adjustable (=1) or fixed (=0) timestep.
IPOLL	Transport model (1 on) [ONLY VALID FOR THE TRAPPA VERSION]
ISPEC	Number of species (=0 : TRAPPA off)
IIRIS	Selection of the used radiation model =0, Semanu =1, IRIS
ICLOUD	Not yet implemented
IART	Additional numerical diffusion =0, no additional numerical diffusion; =1, full numerical diffusion; =2, RSBD
IPHI	Correction factor of the k-model, depends on the solar radiation =1, it is switched on; =2, it is switched off
INBOUS	Switch, if the calculation run with a Boussinesq-model or not =0, no Boussinesq; =1, with Boussinesq
INFRIC	Here it can be chosen between a calculation with or without friction on the surface =0, with friction; =1, without friction
INCORI	With this parameter the coriolis-force can switched on or off for the calculation =0, switched on; =1, switched off
INQSCA	Query if the condensation-scheme is switched on or off =0, switched on; =1, switched off
IGAUSS	Selection between different methods to calculate the vertical turbulence =0, explicit method; =1 semi implicit method (higher precision, but the calculation lasts longer)
IBOTT	Query if the ground-pressure shall be iterated or not =0, no iteration; =1, iteration (with this selection you reach a higher precision in the results, but the calculation lasts longer)
IPTOI	With this parameter it can be defined whether the ground-level air-temperature will be calculated or read from a file =0, MEMO calculate the temperature itself (with this selection you reach a higher precision in the results, but the calculation lasts longer) =1, data will be read from a file
IBCIM	At drastic weather-changes, the lateral boundary conditions must be changed with time. The new values, which are defined in the meteorological-file, will be set to the new boundary conditions (Dirichlet) at the defined time . =0, constant boundary conditions; =1, time-dependent boundary conditions

IRBU	<p>This parameter defines the type of the lateral boundary conditions for the x-component u of the velocity (4 values for each side).</p> <p>You can choose between three types of boundary conditions (The BC's are calculated in the subroutine bctorl.f90).</p>							
	<p>0 = Radiation-BC 1 = Dirichlet-BC 2 = Staggered Neumann-BC</p>	<p>The first number is the definition of the I=1-side, the second of I=IM+2-side.</p> <p>The third number is the definition of the J=1-side, the fourth of J=JM+2-side.</p> <table border="1" data-bbox="580 524 1442 1346"> <tr> <td data-bbox="580 524 799 618">Dirichlet-BC:</td> <td data-bbox="804 524 1442 618">Definition of the BCs according to Dirichlet, means the values of each variable are fixed.</td> </tr> <tr> <td data-bbox="580 624 799 714">Staggered Neumann-BC:</td> <td data-bbox="804 624 1442 714">With BCs according Neumann, the gradient of the variables will set to zero.</td> </tr> <tr> <td data-bbox="580 721 799 1346">Radiation-BC:</td> <td data-bbox="804 721 1442 1346"> <p>That is a special type of boundary conditions. This type allows waves to leave the calculation area with a minimum of reflection. If a different type of BC's are used, the calculation area must be chosen wide and high enough, so as to minimise the influence of the disturbance on the part which is important for the investigation.</p> <p>In the numerical implementation of the radiation-BC, Dirichlet-BCs are calculated for each time step, which are depend on the dispersion of the variables in the calculation area as well as on the dispersion of the variables in the enclosing area. If this type of BC is used for the normal components of velocity, then, for consistency reasons, the homogeneous Neumann-BC should be used for the BC of the non-hydrostatic-pressure.</p> </td> </tr> </table>	Dirichlet-BC:	Definition of the BCs according to Dirichlet, means the values of each variable are fixed.	Staggered Neumann-BC:	With BCs according Neumann, the gradient of the variables will set to zero.	Radiation-BC:	<p>That is a special type of boundary conditions. This type allows waves to leave the calculation area with a minimum of reflection. If a different type of BC's are used, the calculation area must be chosen wide and high enough, so as to minimise the influence of the disturbance on the part which is important for the investigation.</p> <p>In the numerical implementation of the radiation-BC, Dirichlet-BCs are calculated for each time step, which are depend on the dispersion of the variables in the calculation area as well as on the dispersion of the variables in the enclosing area. If this type of BC is used for the normal components of velocity, then, for consistency reasons, the homogeneous Neumann-BC should be used for the BC of the non-hydrostatic-pressure.</p>
Dirichlet-BC:	Definition of the BCs according to Dirichlet, means the values of each variable are fixed.							
Staggered Neumann-BC:	With BCs according Neumann, the gradient of the variables will set to zero.							
Radiation-BC:	<p>That is a special type of boundary conditions. This type allows waves to leave the calculation area with a minimum of reflection. If a different type of BC's are used, the calculation area must be chosen wide and high enough, so as to minimise the influence of the disturbance on the part which is important for the investigation.</p> <p>In the numerical implementation of the radiation-BC, Dirichlet-BCs are calculated for each time step, which are depend on the dispersion of the variables in the calculation area as well as on the dispersion of the variables in the enclosing area. If this type of BC is used for the normal components of velocity, then, for consistency reasons, the homogeneous Neumann-BC should be used for the BC of the non-hydrostatic-pressure.</p>							
IRBV	The same definition as in IRBU, here for the y-velocity-component v.							
IRBW	The same definition as in IRBU, here for the z-velocity-component w.							
IRBT	The same definition as in IRBU, here for the temperature.							
IRBQ	The same definition as in IRBU, here for the humidity.							
IRBX	Definition of the non-hydrostatic-pressure-BC on the lateral sides. Here in x-direction.							
	<p>1 = D -D 2 = NS - NS 3 = D -</p>	<p>D = Dirichlet-BC NS = Staggered Neumann-BC</p> <p>The first specification refers to the side in front of the calculation area while the second one refers to the backside.</p>						

	NS 4 = NS - D	while the second one refers to the backside. If the radiation-BC is used for the normal components of velocity at the lateral boundaries, then, for consistency reasons, the homogeneous Neumann-BC should be used for the BC of the non-hydrostatic-pressure.
IRBY	The same definition as in IRBX, here for the y-direction.	
IRBZ	Definition of the BC of the non-hydrostatic-pressure on the upper horizontal ($k = K+0.5$). The BCs, for the upper boundary are calculated in the subroutine bctorl.f90.	
	1 = D 2 = DS 3 = NS	DS = Staggered Dirichlet-BC For the non-hydrostatic-pressure the common value for this boundary is 2 ($p_{nh}(i,j,k)+p_{nh}(i,j,k+1)=0$). This simple form of boundary condition is justified, because the vertical velocity component is small at the upper boundary of calculation area, therefore the non-hydrostatic effects are negligible. The other reason to use Dirichlet-BC is that homogeneous Neumann-BC are used at the lateral boundaries for the non-hydrostatic-pressure, which means that the use of Neumann-BC at the upper boundary of calculation area would lead to a singular elliptical difference equation.
IRBZTO	Definition of the BC of scalar variables (temperature, humidity, turbulent kinetic energy and dissipation, concentration) on the upper horizontal boundary. The definition of the BC-types is the same as under IRBZ. Normally you choose for this parameter BC-type number 3, in which case homogeneous Neumann-BC will be set for the turbulent kinetic energy and for the temperature, inhomogeneous Neumann-BC for the humidity and TRAPPA concentrations.	
IRBZTU	Definition of the BC of temperature on the lower horizontal boundary. The definition of the BC-types is the same as under IRBZ. Normally you choose for this parameter BC-type 2. This effected that the temperature can be change with the height.	
IRBZUO	Definition of the BC's for the velocity on the upper horizontal boundary.	
	= 2	The velocity changes with the height at the upper horizontal boundary (Staggered Dirichlet-BC).
	= 3,	That means the gradient of velocity is equal zero, thus there is no change in the vertical velocity with the height (homogeneous Neumann-BC).
IRBZUU	Definition of the BC's of the velocity on the lower horizontal boundary.	
	< 3	The velocity changes with the height at the lower horizontal boundary (Staggered Dirichlet-BC). It means, there is a friction on the ground surface.
	= 3,	That means, the gradient of the velocity is equal zero, thus there is no change in the vertical velocity with the height (homogeneous Neumann-BC). It also means, there is no friction on the ground surface.

ITEMAX	The maximum number of iteration steps to calculate the non-hydrostatic-pressure in the elliptical solver. The bigger the number, the more exact is the result, but also the longer the calculation. A good value for this parameter is 50.
PI	This is the number π .
G1	Initialization value for an Adams-Bashforth-coefficient. This parameter needn't be changed.
G2	Initialization value for an Adams-Bashforth-coefficient. This parameter needn't be changed.
ZBO	Reference height to calculate the reference pressure.
XKTH	Horizontal exchange coefficient for scalar variables.
XKMH	Horizontal exchange coefficient for momentum.
XKTMIN	Minimal vertical exchange coefficient for scalar variables.
XKMMIN	Minimal vertical exchange coefficient for momentum.
SECEM	This is an inactive setting
RTOP	This is a parameter that has to do with the initialisation of hydrostatic pressure, its value doesn't ever have to be changed.
UNIT	Allocation of several numbers of Fortran units. Meteorology = 2; Recover = 3; User Data = 5; Logfile = 7; Topography and Landuse = 10
TBCTIM	Control the timer for reading the boundary conditions and for time-dependent BC interpolation, normally only the default values are used.
TUPD	Control the timer for reading the boundary conditions and for time-dependent BC interpolation, normally only the default values are used.
IUPD	Control the timer for reading the boundary conditions and for time-dependent BC interpolation, normally only the default values are used.
TADD	Reference value for the sea-level default temperature during initialisation. tadd is also used in various points in the model to convert between Celsius and Kelvin temperature scales.
P0	Reference value for the sea-level hydrostatic pressure during initialisation.
GAMGEO	Geothermal temperature gradient [$^{\circ}\text{C}/\text{m}$]

SIG	Stefan-Boltzmann-Constant
GRAVITY	Value of the gravity
R_L	Specific gas constant for air.
CPL	Heat capacity of the air.
REL	The value of the relative humidity at the beginning of the calculation.

An example of the Configuration file [config.dat] is given below:

```

!----- Data file -----
!
!           Institut fuer Technische Thermodynamik
!           Universitaet Karlsruhe
!           Kaiserstr. 12
!           D-76128 Karlsruhe
!           Germany
!
!----- Data file -----
!
! Purpose
!
! All physical and numerical constants are given here.
! Additionally default switches are determined.
!
!-----
!       1. Part: Switches to change
!-----

!.. Switch whether (1) or not (0) non-hydrostatic
INHYD=    1

!.. Switch which method to calculate hydrost. pressure
!       (=1:HYD3NL, =2: HYD4NL, ELSE:HYD3)
IHYDNL=   1

!.. Switch which turbulence model
!       (=1:PAND., =2:SCHUM., =3:K-MOD, =4:K-EPS-MOD, <0:TTT)
ITURB=    3

!.. Switch which advection scheme
!       (=1: TVD, =2: FCT)
IADV=     1

!.. Switch for adjustable (1) or fixed (0) time increment
IAUTO=    0

!.. Switch for Transport Model (=1 on, =0 off)
IPOLL=    0

!.. # of Species in Emissions Inventory
ISPEC=    3

!.. Switch for Radiation Model (=1 IRIS, =0 SEMANU)
IIRIS=    1

```

```

!.. Switch for Cloud Model (not yet implemented)
ICLOUD= 1

c.. Switch for artificial diff. (=0 OFF, =1 FULL, =2 RSBD)
IART= 2

c.. Switch whether (1) or not (0) PHI-correcture
IPHI= 1

c.. Switch whether (1) or not (0) non-bousinesq
INBOUS= 0

c.. Friction (0) or not (1) at the surface
INFRIC= 0

c.. Coriolis force (0) or not (1) considered
INCORI= 0

c.. Condensation (0) or not (1) considered (to be changed)
INQSCA= 1

c.. Semi-implicite (1) or explicite (0) treatment of vertical turbulence
IGAUSS= 1

c.. Iteration (1) or not (0) of the surface pressure
IBOTT= 1

c.. Initial near ground air temperature results from T0I (1) or not (0)
IPT0I= 0

c.. Switch for lateral BC to be timedep. (1) or not (0)(to be changed)
IBCTIM= 1

!-----
! 2. Part: boundary switches
!-----
IRBU= 0 0 0 0
IRBV= 0 0 0 0
IRBW= 0 0 0 0
IRBT= 0 0 0 0
IRBQ= 0 0 0 0

IRBX= 2
IRBY= 2
IRBZ= 2

IRBZTO= 3
IRBZTU= 2
IRBZUO= 3
! IRBZUU = 3: d(u,v)/dz = 0 ==> no friction at the surface
IRBZUU= 2

!-----
! 3. Part: Initialization of numerical quantities
!-----
!.. Max. # of iterations in NHYD3
ITEMAX= 50

!.. Simply PI
PI= 3.1415927

!.. Initial values for Adams-Bashforth-coefficients
G1= 1.5
G2= -0.5

!.. Base height for calculation of base state pressure
ZB0= 0.

```

```

!.. Constant artificial diffusivity for scalars and momentum
XKTH=    100.
XKMH=    100.

!.. Minimum value for eddy viscosity and diffusivity
XKTMIN=  0.5
XKMMIN=  0.5

!.. Initialization of emission time control (to be changed)
SECEM=   0. 0. 0.

!.. Initialization of RTOP
RTOP=    0.5

!-----
!   4. Part: Initialization of FORTRAN UNIT numbers
!-----

!.. Meteorology           2
!.. Recover               3
!.. User data              5
!.. Logfile                7
!.. Topography and landuse 10+IN

!..          LMET, LCASE, LLOG, LTOP
UNIT=        2,    3,    7,    10

!-----
!   5. Part: Initialization of Timers
!-----

TBCTIM=  0.
TUPD=    0.
IUPD=    0

!-----
!   6. Part: Initialization of physical quantities
!-----

!.. Ref. value for base state pressure (100 MPa)
TADD=    273.16

!.. Ref. value for base state pressure (100 MPa)
P0=      1.E5

!.. Geothermal temperature gradient (used for initial. of T2B)
GAMGEO=  -6.5E-3

!.. Stefan-Boltzmann constant
SIG=     5.67E-8 /

!.. Gravity acceleration, specific gas constant and heat capacity of air,
GRAVITY= 9.81
R_L=     287.04
CPL=     1.006E3

!.. Relative humidity at the begin of a run
REL=     0.45

!.. Compositions of it
!   GRL = G/RL
GRL=     0.0341764
!   GCP = G/CPL
GCP=     0.0097514
!   CPR = CPL/RL
CPR=     3.504738
!   RCP = RL/CPL

```

RCP= 0.285328

!.. Quantities needed for the radiation budget

SOAB= 0.5
SOSC= 0.2
XNO= 0.0
VOZON= 0.3
UTWUP= 0.0
CPOL= 7.8737E-9
CCO2= 320.E3

5. The Meteorology file

In the TEMPOL and VELPOL subroutines the following meteorological data is read from the Input Meteorological (soundings) file:

In TEMPOL:

```

10      READ (40,*) ABG
        READ (40,*) XBG(I), YBG(I), TBG(I), ZSBG(I)
        CONTINUE

        READ (40,*) GAMT, ZINP

        READ (40,*) ATG, IFTG
        DO 11 I=1,ATG
        READ (40,*) NTG(I), XTG(I), YTG(I), ZSTG(I)
        DO 11 J=1,NTG(I)
        READ (40,*) TTG(J,I), ZTG(J,I)
11      CONTINUE

```

In VELPOL:

```

10      READ (40,*) AMG
        DO 10 I=1,AMG
        READ (40,*) XMG(I), YMG(I), UMG(I), VMG(I), ZMG(I)
        CONTINUE

        READ (40,*) ASG, IFSG
        DO 20 I=1,ASG
        READ (40,*) NSG(I), XSG(I), YSG(I), ZSSG(I)
        DO 20 J=1,NSG(I)
        READ (40,*) USG(J,I), VSG(J,I), ZTG(J,I)
20      CONTINUE

```

Variable	Type / Dimension	Unit	Meaning
ABG	Integer / 1	-	Number of temperature measuring stations on the surface < 0: potential temp. as input
XBG YBG	Real / ABG	m	Distance between gird origin and surface temperature measurement station, x/y-direction
TBG	Real / ABG	C	Air temperature at the surface
ZSBG	Real / ABG	m	Surface height at the station location
GAMT	Real / 1	C/m	Constant temperature gradient in the upper air
ZINP	Real / 1	m	Height(ASL) where temperature starts to become only dependant on z-coordinate
ATG	Integer / 1	1	Number of upper air sounding locations

			> 0: absolute temperature as input < 0: potential temp. as input (both in Celsius)
IFTG	Integer / 1	1	= 0: ZTG is height above sea level = 1: ZTG is height above surface
NTG	Integer / ATG	1	Number of temperature soundings for a certain location
XTG YTG	Real / ATG	m	Distance between origin and upper air temperature sounding location, x/y-direction
ZSTG	Real / ATG	m	Surface height at upper air temperature sounding location
TTG	Real / ATG,NTG	C	Sounded temperature
ZTG	Real / ATG,NTG	m	Height of temperature sounding
AMG	Integer / 1	-	Number of wind measuring stations at the surface
XMG YMG	Real / AMG	m	Distance between origin and surface wind measuring, x/y-direction
UMG VMG	Real / AMG	m/s	Wind component of surface measuring x/y-direction
ZMG	Real / AMG	m	Height of wind measuring station
ASG	Integer / 1	-	Number of upper air wind soundings
IFSG	Integer / 1	-	= 0: ZSG is height above sea level = 1: ZSG is height above surface
NSG	Integer / ASG	-	Number of wind soundings at a certain location
XSG YSG	Real / ASG	m	Distance between origin and upper air wind sounding location, x-direction
ZSSG	Real / ASG	m	Surface height at upper air wind sounding location
USG VSG	Real / ASG,NSG	m/s	Sounded wind component, x/y-direction
ZSG	Real / ASG,NSG	m	Height of wind sounding
CKEY	Character*60	-	Key word
LTIME	Integer / 1	hh mm	Time since simulation start

An example of the Meteorology input file is given here (only the header section is shown):

6. The Path-description file

In this configuration file, various file names for input and output must be specified. Note that a separate topography file name should be specified for every nesting level. An example path-description file [path.fil] is given below:

```
!
!       This is the file containing the I/O filenames for MEMO 6
!       The file has a variable structure
!       The # of nests is given by the number of topography files
!
!-----
!
!       1) Common for all Nests
!
!       Configuration file
ConfigFile=.config.dat
!
!       File specifying the case
CaseFile=info.cas
!
!       Meteorology file
MeteorologyFile=soundnew05.1.met
!
!       Logfile
LogFile=logfile.dat
!
!       2) Repeated for each Nest
!
!       Topography file
TopographyFile=topomemo_72x72_4.dat
TopographyFile=topomemonest_72x72_1.dat
```

7. Shell Scripts

The two scripts included in this chapter are normally used in Unix to invoke the preprocessor, the main model and the postprocessor executables.

7.1 The "low level" script

This script can be invoked (in Unix) as follows:

```
./m_execute [options]
```

Where [options] can be one or more of the following:

- a : Use ASCII storage mode for the input and output of recovery files (if omitted, BINARY mode is used)
- p : invoke the preprocessor (prep6 or MEMOPREP.exe)
- e : invoke the main executable (memo6 or MEMOMAIN.exe)
- m : invoke the MIMO coupling executable [not applicable for the current version]
- c : invoke the plot-from-recovery post-processor (plot6 or MEMOPLOT.exe)

In order for the .m_execute script to function properly, various "environment variables" need to be set before its execution:

- RECINI : "output" recovery file name for the preprocessor, "input" recovery file name for the main executable
- RECOUT : "output" recovery file name for the main executable
- INFILE : filename for the path-description file
- PLOTFI : base filename for the main Output files (.1 or .2 is automatically appended to this filename, in order to indicate the nest)
- COUPLE : filename for the MEMO-MIMO coupling file [inactive in this version]
- COUPL_IF : MIMO files, not applicable in this version
- COUPL_CONT : MIMO files, not applicable in this version

Normally, all those environment variables are appropriately set by the second, "high level" script, before calling .m_execute.

7.2 The high-level script

This script is simply invoked (in Unix) as follows:

```
./runcase
```

(no arguments)

In DOS, the corresponding RUNCASE.BAT script simply sets the necessary environment variables and then directly calls the preprocessor (MEMOPREP.exe) and the main executable (MEMOMAIN.exe). If one would like to call only the main executable, it would be easy to comment-out (by prefixing the relevant lines with "REM") the call to the preprocessor, or use a different script altogether.

Please note that the base name of the MEMO output files is only specified in this script.

7.3 Implementation of the scripts (Unix)

.m execute (driver script):

```
#!/bin/sh
#-----
#
#   Execute script
#
#-----
# Check command line args
for i; do
  if test $i = -a; then      M_ASC=on; fi # ASCII/BINARY
  if test $i = -h; then      M_HEL=on; fi # Help mess.
  if test $i = -p; then      MODE=PRE; fi # Preprocessor
  if test $i = -e; then      MODE=MAI; fi # Main run
  if test $i = -m; then      MODE=MIM; fi # Coupling with MIMO
  if test $i = -c; then      MODE=PLO; fi # Plot from recover
done
if test -z "$M_ASC"; then M_BIN=-b; fi

if test -n "$M_HEL"; then
  echo 'usage execute [opts] <file containing filenames>'
  echo 'opts:  -a  ASCII output on recover file'
  echo '          -p  Preprocessor run'
  echo '          -e  Main run'
  echo '          -m  Coupling with MIMO'
  echo '          -c  Plot from recover'
  exit
fi

if test -z "$MODE"; then echo ' No action specified '; exit; fi
if test -n "$M_ASC"; then echo 'ascii      version'; fi

# Distinguish and process action
case $MODE in
  PRE)      # run preprocessor
  echo 'Preprocessor run'
  ./prep6 $M_BIN -f2=$RECINI -f1=$INFILE; ;;
  MAI)      # run main program
  echo 'Main run'
  ./memo6 $M_BIN -f1=$RECINI -f2=$RECOU -fp=$PLOTFI -fm=$COUPLE; ;;
  MIM)      # run mimo postprocessor
  echo 'MIMO Coupling run'
  ./post6 $M_BIN -f1=$COUPL_IF -fc=$COUPL_CONT; ;;
  PLO)      # run plot postprocessor
  echo 'Plot Postprocessor run'
  ./plot6 $M_BIN -f1=$RECINI -fp=$PLOTFI; ;;
esac
```

runcase (high-level execute script):

```
#!/bin/sh
#-----
#
#   Execute script
#
#-----
# Update executables
#   ./m_compile -o
# Preprocessor run
  export INFILE=path.fil
```

```
export RECINI=OUTfiles/recover.in
./m_execute -p -a

# Main run
export RECINI=OUTfiles/recover.in
export RECOUT=OUTfiles/recover.out
export FSTARTME=OUTfiles/test.startme
export FEND=OUTfiles/test.end
export PLOTFI=OUTfiles/CITY2002_may08
export COUPLE=OUTfiles/couple
./m_execute -e -a
```

7.3 Implementation of the driver script (DOS/Windows)

RUNCASE.BAT (high-level execute script):

```
set infile=path.fil
set recini=OUTfiles\recover.in

echo 'Preprocessor run'
MEMOPREP.exe -b -f2=%recini% -f1=%infile%

set plotfi=OUTfiles\CITY2002_may08
set recout=OUTfiles\recover.out
echo 'Main run'
MEMOMAIN.exe -b -f1=%recini% -f2=%recout% -fp=%plotfi%
```

8. Description of the ITT 2000 format

MEMO uses the ITT 2000 format for storing 2D and 3D fields of vector and scalar quantities. One separate file is stored for each nesting level.

The file contains a header and one or more time-stamped sections. Each section contains several subsections, one for each scalar 2- or 3-D field or for each one of the three components of a vector 2- or 3-D field.

3-D fields are stored as consecutive 2-D blocks, each 2-D block corresponds to a z-section of the field. So, an output file for a 3D vector field (e.g. wind speed), and two 3D scalar fields (e.g. temperature, absolute humidity) would have the following structure:

```
[File Header]

[Section: t=0.0 sec]
  [Subsection: Wind Speed, U component]
    [2D block for Z=0]
    [2D block for Z=1]
    ....
  [Subsection: Wind Speed, V component]
    [2D block for Z=0]
    [2D block for Z=1]
    ....
  [Subsection: Wind Speed, W component]
    [2D block for Z=0]
    [2D block for Z=1]
    ....
  [Subsection: Temperature]
    [2D block for Z=0]
    [2D block for Z=1]
    ....
  [Subsection: Abs. Humidity]
    [2D block for Z=0]
    [2D block for Z=1]
    ....
[Section: t=3600.0 sec]
  [Subsection: Wind Speed, U component]
    [2D block for Z=0]
    [2D block for Z=1]
    ....
```

etc.

8.1. The File Header

The format of the file header is as follows:

<u>Line Format</u>	<u>Description</u>
lines 1-3:	Three comment Lines
<i0>, <j0>, <k0>	first values (0 or 1) for each of the three indexes, i,j,k
<i1>, <j1>, <k1>, <dx>, <dy>, <dz>, <maxz>	last values for each of the three indexes, cell size (increment) and maximum extent of the grid in

the z-direction.
 Iff $\langle dx \rangle, \langle dy \rangle, \langle dz \rangle \leq 0$, then UTM coordinates for each grid point are also stored in the header.

next ? Lines:

x- (UTM easting) positions of the grid lines.

next ? Lines:

y- (UTM northing) positions of the grid lines.

next ? Lines:

z- height above surface, in meters of the grid levels.

next ? Lines:

topographic heights at each grid point, in meters.

0.0 0.0 0.0

This line is not used

$\langle NV3D \rangle, \langle NS3D \rangle, \langle NV2D \rangle, \langle NS2D \rangle, \langle NS1D \rangle, \langle NPF \rangle, \langle NTS \rangle, 0, 0, 0$

These values normally indicate the number of 2D/3D scalar/vector subsections stored in each section.

8.2. Section Header

Each section contains the following header:

Line Format	Description
$\langle dd \rangle. \langle mm \rangle. \langle yy \rangle \quad \langle hhmm \rangle \quad LST \quad \langle time2 \rangle \quad 99 \quad \langle num \rangle \quad 0$	Model Date and Time, $\langle time2 \rangle$ is model time in hours, $\langle num \rangle$ is the number of the current section.

8.3. Subsection Header

Each subsection contains the following header:

Line Format	Description
$\langle description \rangle \quad \langle symbol \rangle \quad \langle unit \rangle$	Description of the variable, physical symbol, units.
$\langle i0 \rangle \quad \langle i1 \rangle \quad \langle j0 \rangle \quad \langle j1 \rangle \quad \langle k0 \rangle \quad \langle k1 \rangle \quad 3 \quad \langle iart \rangle \quad \langle ideof \rangle \quad 0 \quad \langle dt \rangle$	$\langle i0 \rangle$ - $\langle i1 \rangle$: extent of the i-index $\langle j0 \rangle$ - $\langle j1 \rangle$: extent of the j-index $\langle k0 \rangle$ - $\langle k1 \rangle$: extent of the k-index $\langle iart \rangle$: type of field indicator $iart=1$: 3D-vector component $iart=2$: 2D-scalar field $iart=4$: 2D-scalar field $\langle ideof \rangle$: vector component index $ideof=1$: u-component $ideof=2$: v-component $ideof=3$: w-component

<dt>: model timestep (in sec)

8.4. 2D Block

Each 2D block contains the following:

<u>Line Format</u>	<u>Description</u>
<a> <min> <max>	<a> and are the scaling factors for obtaining the real values from the integer-encoded values on the data block. <min> and <max> are the minimum and maximum values of the variable for this 2D block.
next ? Lines	Integer-encoded values (see below)

In each 2D block, $(i1-i0+1)*(j1-j0+1)$ integer-encoded values are stored, each ranging from **-980 to 9980**. These values represent the array $(i0:i1, j0:j1)$ of (real) variable values on the corresponding grid points. The fast-changing index (i) corresponds to the x-direction, the slow-changing one (j) to the y-direction.

For each integer-encoded value ii of the block, the corresponding true (real) value, ff , of the variable can be calculated as:

$$ff = (ii - b) * a$$

8.5. 3D Block

Each 3D block contains $(k1-k0+1)$ consecutive 2D block, each of which is preceded by a separate <a> <min> <max> header, i.e. each 2D block can have different scaling factors. The 2D blocks are stored with increasing k-index.

9. Recovery files

"**recover.in**" is the "recovery" file, containing a snapshot of all relevant variables and fields needed to initialize or continue a MEMO run. The preprocessor creates an initial condition for the model, then stores the model state in "**recover.in**".

"**recover.out**" is the recovery output file for the main run, stored only at the end of the simulation.

A recovery run is possible only for continuing the simulation after a successful termination of the main executable. Of course, it would be easy to change the code of MEMO to periodically store recovery snapshots, so as to be able to continue interrupted runs, but currently this is not implemented (except for coupling with MIMO, but the code is presently deactivated).

For a recover (continuation) run, one would rename **recover.out** to **recover.in**, change the time and date indicators in the Case file, and then manually execute **memo6 (MEMOMAIN.exe)** (without calling the preprocessor) with the appropriate command-line arguments, as indicated in the second part of the **runcase (RUNCASE.BAT)** script (section 7).

10. Converter template for the ITT format

The program ITTTEST demonstrates the extraction of time-series of scalar and vector data from the ITT output files of MEMO. The main program module can be easily modified in order to e.g. obtain complete field snapshots, instead of time series, in the appropriate format for external visualisation programs.

ITTTEST stores time-series of user-selected variables at various “station locations”, specified using i- and j- grid coordinates. Additionally, a unique name should be defined for each station/location.

The invocation tree of ITTTEST has the following structure:

```
PROGRAM ITTTEST (ittttest.f)           ! Main program, parameter definitions
|
|--SUBROUTINE INHEAD (inhead.f)       ! Reading of the ITT header
|
*--SUBROUTINE INMAIN (inmain.f)       ! Reading of the rest of ITT output
|
|   |--SUBROUTINE PLOTR (plotr.f)      ! Reading of a 2D or 3D array
|   |
|   |   *-- SUBROUTINE PREAD (pread.f) ! Reading of a 2D or 3D array
|   |
|   |   *-- SUBROUTINE CONVERT (convert.f) ! Conversion of grid points
```

Compilation and installation of ITTTEST can be performed using the included Makefile and the GNU make utility (if available on the user's system). Alternatively, all of the six source (*.f) files could be imported as a single project in a visual development environment (e.g. Visual Fortran for Windows) and compiled together to obtain the ITTTEST.exe executable.

The main executable is simply invoked, without any command-line arguments, as:

```
./ittttest           (in Unix/Linux/AIX)
ITTTEST.exe         (in DOS/Windows)
```

after which a message appears, prompting the user to input the **filename of the input ITT file** (e.g. "test.1"):

```
ITT file name : test.1[ENTER]
```

This is the only user input during run-time. The rest of the configuration options must be specified in the source of the main program (ittttest.f).

For non-interactive invocation of ittttest, the following format could be used instead (in Unix):

```
echo test.1 | ./ittttest
```

10.1 Important user-settable parameters in ittttest.f

The following is an example of station-related parameters set in ittttest.f:

```
INTEGER, PARAMETER :: NSMAX = 2           ! Number of stations/locations for
                                           ! time-series output (here 2).
DATA      KSTAX/ 1,15/                    ! i- coordinates for all the stations.
DATA      KSTAY/ 2,15/                    ! j- coordinates for all the stations.
DATA      OUTLAYER/ 2, 2/                 ! k- coordinate (z-layer) of output
                                           ! for each station.
DATA      CSTA/'STATION1',
           'STATION2' /                   ! Unique ID for each station.
```

Using the above settings, the program would generate two output files, named `st_STATION1.DAT` and `st_STATION2.DAT`, corresponding to locations $(i=1,j=2,z=2)$ and $(i=15,j=15,z=2)$ of the MEMO grid. Each file would contain values, in column format, of various prognostic variables for all the timesteps contained in the input ITT file. The format of `st_*.dat` files is described in the next section:

10.2 Output format of ITTTEST

Presently, the output files of ITTTEST consist of eight columns of values, with the number of rows being equal to the number of timesteps in the original ITT file.

Column	Description
1	Line number
2	Date, in dd.mm.yy format
3	Hour of the day, in hhmm format
4	Wind speed, in m/s
5	Wind direction, in degrees
6	Potential Temperature, in degrees Celsius
7	Absolute Humidity, in g/m^3
8	Shortwave Incoming Radiation, in W/m^2

11. The Plotter (plot6) executable

The plot6 (MEMOPLOT.exe) executable extracts plot output from recovery files (see section 9) and stores it in ITT 2000 format (see section 8). This feature can be used e.g. to extract 2D or 3D fields from an initialisation recov.in, generated by the preprocessor, for visual inspection and visualisation. Processing of ITT output of plot6 can be performed by the same tools used to process the output of the main MEMO executable (see section 10).

The plotter executable can be invoked as follows:

```
./plot6 [-b] -f1=<inputfile> -fp=<outputfile>
```

Where:

-b	if specified, selects binary input from the recovery file. By default, the recovery file is assumed to be in ASCII format.
<inputfile>	is the name of the recovery file
<outputfile>	is the name of the output (ITT) plot file.