



Max-Planck-Institut  
für Meteorologie

Max-Planck-Institut für Meteorologie  
Bundesstr. 53  
D-20146 Hamburg



Deutscher Wetterdienst  
Wetter und Klima aus einer Hand

Deutscher Wetterdienst  
Frankfurter Str. 135  
D-63067 Offenbach

# ICON User's Guide

S. Rast

assembled and edited by

T. Schad, I. Kraut, D. Rieger, C. Walter, S. Gruber, K. Deetz, H. Vogel, and B. Vogel



**Karlsruhe Institute of Technology**  
Institute for Meteorology and Climate Research (IMK-TRO)  
Hermann-von-Helmholtz-Platz 1  
D-76344 Eggenstein-Leopoldshafen

August 12, 2016

## Preface

This user guide was assembled and edited based on available documents on the ICON webpage by the persons mentioned at the front page. The content of the user guide follows the requirements of DWD.

*Important hints:*

In chapter 4 a list of the namelist parameters is given. New and inexperienced users should only modify the namelist parameters that are given in bold letters.

When results produced with ICON are published the following papers have to be cited in the list of references:

?

**Information for authors:**

**Please read the README for further instructions and templates.**

# Contents

# 1 Guide for New Users

This tutorial is meant for people with some knowledge and/or experience in modelling and Linux, but which have no experience with the ICON model. In the following we will describe in short how to compile and run ICON on your machine.

## 1.1 Needed Software

For some components ICON uses external libraries. Therefore you will need some additional software which should be installed on your machine. The following software needed to be installed on your machine:

- NetCDF: NetCDF is a set of software libraries and self-describing, machine-independent data formats that support the creation, access, and sharing of array-oriented scientific data.  
(Source: <http://www.unidata.ucar.edu/software/netcdf/>)
- GRIB: GRIB (GRIdded Binary) is a format defined by the WMO (World Meteorological Organization). The use of GRIB in ICON is optional. The ECMWF GRIB API is an application program interface accessible from C, FORTRAN and Python programs developed for encoding and decoding WMO FM-92 GRIB edition 1 and edition 2 messages. A useful set of command line tools is also provided to give quick access to GRIB messages. ICON requires GRIB2 format.  
(Source: <https://software.ecmwf.int/wiki/display/GRIB/Home>)
- MPI: MPI is a library specification for message-passing, proposed as a standard by a broadly based committee of vendors, implementors, and users.  
(Source: <http://www.mcs.anl.gov/research/projects/mpi/>)
- OpenMP: Jointly defined by a group of major computer hardware and software vendors, the OpenMP API is a portable, scalable model that gives shared-memory parallel programmers a simple and flexible interface for developing parallel applications on platforms ranging from embedded systems and accelerator devices to multicore systems and shared-memory systems.  
(Source: <http://openmp.org/wp/>)

## 1.2 The Source Code

You can obtain the source code on the website of DKRZ:

<https://www.dkrz.de/>

You can use the following commands to untar the ICON source code:

```
tar xfvz icon.tar.gz
```

This will create a folder `icon-1.0` inside your current directory. Within the ICON User Guide, this folder will further on be called `$ICONDIR`.

### 1.2.1 Directory structure

Within `$ICONDIR`, you will find a set of subdirectories. The important subdirectories are described in the following.

#### **build**

Within the `$ICONDIR/build` directory, a subdirectory with the name of your computer architecture is created at compilation. Within this subdirectory, a `bin` subdirectory containing the binary `control_model` and several further subdirectories containing the compiled module files are created at compilation.

#### **config**

Inside the `$ICONDIR/config` directory, different machine dependent configuration are stored within the configuration files. You can find a description of how to use and set up such configuration files in chapter ??.

#### **data**

Within the `$ICONDIR/data` directory, you will find divers input datasets. For example, there are the datasets `"rrtmg_lw.nc"` and `"ECHAM6_CldOptProps.nc"`, which are necessary for the radiation scheme (see sec. ??).

#### **doc**

Within the `$ICONDIR/doc` directory, several documentations for ICON are stored. There are according subdirectories for scientific (`$ICONDIR/doc/science`), technical (`$ICONDIR/doc/technical`) and programming style guides (`$ICONDIR/doc/style`).

#### **externals**

Within the `$ICONDIR/externals` directory, external libraries for ICON are stored. Currently, it is the `mtime` library which is used to convert different date time formats.

### **include**

Within the `$ICONDIR/include` directory, interfaces to libraries needed by ICON are stored. Currently, the interface to the CDI library is stored inside this directory.

### **run**

Within the `$ICONDIR/run` directory, namelist descriptor files as well as the full namelist documentation are stored. The namelist descriptor files can be used to generate runscripts. Further information can be found in ??.

### **src**

Within the `$ICONDIR/src` directory, the source code of ICON including the main program and ICON modules can be found. The modules are ordered in several subdirectories which are described in the following.

The main program `control_model.f90` can be found inside the subdirectory `$ICONDIR/src/drivers`. Additionally, this directory contains the modules for a hydrostatic and a nonhydrostatic setup.

The configuration of an ICON run is performed within the modules inside `$ICONDIR/src/configure_model` and `$ICONDIR/src/namelists`. Modules regarding the configuration of idealized test cases can be found inside `$ICONDIR/src/testcases`.

The dynamics of ICON are inside `$ICONDIR/src/atm_dyn_iconam` and the physical parameterizations inside `$ICONDIR/src/atm_phy_nwp`. Parameterizations for the interactions with the surface can be found inside `$ICONDIR/src/lnd_phy_nwp`.

Shared infrastructure modules for 3-D and 4-D variables can be found within `$ICONDIR/src/shared`. The according routines for 2-D fields (e.g. external parameters) are stored within `$ICONDIR/src/shr_horizontal`.

Modules handling the parallelization can be found in `$ICONDIR/src/parallel_infrastructure`.

Input and output modules are stored in `$ICONDIR/src/io`.

The modules for the grid generator, as described in chapter ?? can be found inside `$ICONDIR/src/grid_generator`.

### **support**

Within the `$ICONDIR/support` directory, the CDI library is stored.

## vertical\_coord\_table

Inside the `$ICONDIR/vertical_coord_tables` directory, information files describing the relation between model layer and height are stored.

## 1.3 Configuration and Compilation

To ease up the compilation a configure-file is provided which should take over the main work. This Autoconf configuration is used to analyze the computer architecture (hardware and software) and set user specified preferences, e.g. the compiler. This preferences are read from `config/mh-<OS>`, where `<OS>` is the identified operating system. Operating systems are listed in the configure-files in `$ICONDIR/config/` with the according files `mh-<OS>`. If your machine is not listed you can add a config-file with your own `<OS>` based on the given `mh-<OS>` files. If different compilers are available, the `mh-<OS>` file may contain a case construct to distinguish them. If your `<OS>` is not recognized but is one of the listed `<OS>` you can invoke the configure file with the according option `--host=$HOST`. Examples for the DWD CRAY system are given in the boxes.

### 1.3.1 Description of the Configuration Files

To add a specific compiler or change your compiler flags, you have to enter the `$ICONDIR/config/mh-<OS>` according to your operating system `<OS>`. For the DWD CRAY, the compiler flags in `mh-linux` look like the following:

CRAY EXAMPLE: Compiler Flags inside `mh-linux`

```
config_compiler=cray
  CC          = cc
FC          = ftn
F77         = "$FC"
FFLAGS      = -v -D__LOOP_EXCHANGE -D__MIXED_PRECISION -Df2cFortran
-e Z -em -hflex_mp=conservative -hfp1 -hadd_paren -r am -Ktrap=divz,ovf
CFLAGS      = -I${GRIB_API}/include -v -Df2cFortran
-DHAVE_CF_INTERFACE -DHAVE_LIBNETCDF -DHAVE_LIBGRIB
-DHAVE_LIBGRIB_API -O3 -D__SVN_VERSION="${SVNVERSION}"
F77FLAGS    = "$FFLAGS"
FCLIBS      = "-v"
GEN_FLAGS   =
FDEBUG      = -g -R abc
OMPFLAG     = -mp
DEFOPT      = -D
DEFCOPT     = -D
MODOPT      = -I
MODDIR      =
;;
```

The `cray`) in this example gives the name of this specific configuration. It can be addressed by a flag at configuration. For this example, the according command to choose this setting would be



`./configure --with-fortran=cray` (see section ??). Like this, you can create your own configuration by adding a new compiler.

CC, FC and F77 are the compiler directives for C-Compiler, FORTRAN2003-Compiler and FORTRAN77-Compiler. The according compiler flags are set via CFLAGS, FFLAGS and F77FLAGS. The variable to set an OpenMP flag is called OMPFLAG. Libraries are set via FCLIBS.

### 1.3.2 Configuring and Compiling the Code

To configure the source code go to \$ICONDIR and give:

```
./configure
./build_command
```

If you want to use another compiler than the default compiler you give:

```
./configure --with-fortran=<compiler>
./build_command
```

where <compiler> is {gcc,nag,intel,pgi,cray}.

```
CRAY EXAMPLE: Configure + Make
./configure --with-fortran=cray}
./build_command
```

Note, that CRAY compiler environment (cce) versions 8.2.x do not work with ICON. The CRAY configuration is expanded to the following:

```
CRAY EXAMPLE: Configuration
ftn -I../module -v -D__LOOP_EXCHANGE -D__MIXED_PRECISION -Df2cFortran -e
Z -em -hflex_mp=conservative -hfp1 -hadd_paren -r am -Ktrap=divz,ovf
-D__ICON__ <object files> -L/usr/local/pkg/grib_api/1.11.0/CRAY/lib
-L../lib -lsupport -lgrib_api_f90 -lgrib_api -lmttime $(LAPACK_LIB)
$(NETCDF_LIB) $(HDF5_LIB) $(SZIP_LIB) $(ZLIB_LIB) $(MPI_LIB)
$(PROFILE_LIB) $(SCT_LIB)
```

ICON is parallelized using MPI and OpenMP. You can control the parallelization to be used by giving:

```
./configure --with-mpi/--without-mpi --with-openmp/--without-openmp
./build_command
```

By default the options are set to `--with-mpi --without-openmp`. After a successful build, you will find the ICON executable named `control_model` inside `$ICONDIR/build/<OS>/bin/`. The CRAY

Fortran compiler is an exception, as the command includes automatically OpenMP. Therefore, although selecting `--without-openmp`, OpenMP is used.

If you wish to re-configure ICON it is advisable first to clean the old setup by giving:

```
make distclean
```

Some more details on configure options can be found in the help of the configure command:

```
./configure --help
```

## 1.4 Running the model (Test scripts for ICON)

### 1.4.1 Principles of testing ICON

The ICON developers use the buildbot tool in order to perform automated tests on selected ICON experiments at regular time intervals. The buildbot tool launches the respective test scripts on various computer platforms and documents success or failure on a special web site (<https://buildbot.zmaw.de/icon>). The automated tests are performed on the newest model revision as available in the ICON repository. Furthermore, tests can be “forced”, i.e. started by hand, at any time specifying a certain experiment and model revision of any branch of the repository. However, it is impossible to test two revisions against each other or to test a local revision. Here, we present new tests for ICON into which various experiments are integrated and which are designed such that they can either be used in the framework of the buildbot tool or be started manually without reference to buildbot (e.g. on a PC at MPI Hamburg). The tests are designed to trap certain technical errors and comprise the following experiments:

**base test:** Just a simple base run over a short time period for a specific experiment. This run will be called simulation A in the following.

**update test:** In addition to the model revision to test, the so-called “test revision”, a “reference revision” can be specified. A short simulation A of the test revision over one hour is performed during which restart files are written. The same simulation is performed for the reference revision (simulation A’). The “update test” is said to be passed if there are no differences in the output of simulation A and A’ using the “cdo diff” command on the time steps in the output specified by the user.

**restart test:** In addition to a base simulation A, a second simulation B restarts ICON at some time after the initial date. The “restart test” is said to be passed if there are no differences in the output for the time steps after the restart between the original and the restarted simulation using the “cdo diff” command.

**nproma test:** The nproma test performs a base simulation A and a simulation C with a different value of `nproma`. Instead of `nproma` of simulation A, a value of 17 is used in simulation C or, if `nproma = 17` for simulation A, `nproma = 19` is used for simulation C. The nproma test is said to be passed if the “cdo diff” command does not find differences in the output.

**mpi parallel test:** The mpi parallel test performs a base simulation A and a simulation D with a reduced number of MPI threads compared to the base simulation A. If more than one threads are used on each node, the number of threads on each node is reduced by one. If only one thread is used on each node, the number of nodes is reduced by one. If only one process is used, no mpi parallel test is performed. The parallel test is said to be passed if the “cdo diff” command does not find differences between the output files.

**openmp parallel test:** The openmp parallel test performs a base simulation A and a simulation E with a reduced number of openmp threads compared to the base simulation A. If only one openmp thread was used, no openmp test is performed. The openmp parallel test is said to be passed if the “cdo diff” command does not find differences between the output files.

The testing procedure is such that tests can be combined. Furthermore, the test script can be asked to re-use existing runs without repeating these runs.

Only the following experiments are included into the test script:

**atm\_amip\_test:** Non-hydrostatic AMIP-like simulation but with transient solar irradiance using ECHAM physics.

**atm\_icoles\_nested:** Nonhydrostatic atmosphere only simulation with a regional grid refinement.

**atm\_jww\_hs\_test:** Jablonowski Williamson baroclinic wave test for a hydrostatic atmosphere.

**oce\_omip\_0160km:** Ocean only experiment with a 160km resolution.

### 1.4.2 Description of test script

The test script `icon-dev.checksuite` is located in the `run/checksuite.icon-dev` directory of ICON and uses the following run script of the `run` directory for the experiments: (i) `exp.atm_amip_test` for the AMIP-type experiment `atm_amip_test`, (ii) `exp.atm_icoles_nested` for the atmosphere experiment with a grid refinement, (iii) `exp.atm_jww_hs_test` for the Jablonowski Williamson baroclinic wave test, and (iv) `exp.oce_omip_0160km` for the ocean only experiment `oce_omip_0160km`.

These run scripts contain all necessary namelist groups and links to files that contain the initial and boundary conditions. By the standard `make_runscripts` command invoked inside `icon-dev.checksuite`, this script is transformed into the actually used form that contains an additional suffix `.run` at the end of its name. **Attention:** `icon-dev.checksuite` generates the specific run script by default and overwrites those that are present. The script can be forced to use present runscripts. For the various test runs for each experiment, these `.run` files are copied and edited by `sed`.

Here follows a more detailed description of the script:

**icon-dev.checksuite:** This script uses the `make_runscripts` command to produce `exp.<exp_name>.run` from the basic run script `exp.<exp_name>`. The default is that any existing run script is overwritten but there is an option to keep existing runscripts. These run scripts are then modified by `sed` commands in order to perform the various test runs. Once the test runs are finished, the function `diff_results` of `icon-dev.checksuite` is called to determine the differences between those runs.

**exp.<experiment>:** These scripts contain all settings for the base simulation in one experiment. To date, the experiments `<experiment> = atm_amip_test, atm_icoles_nested,`

`atm_jww_hs_test`, and `oce_omip_160km` can be used in the tests. The base script `exp.<experiment>` will be transformed by `make_runscripts` into a script that can actually run the ICON model. The resulting `exp.<experiment>.run` scripts will then be copied to `exp.<experiment>_base.run`, `exp.<experiment>_restart.run`, `exp.<experiment>_nproma.run`, `exp.<experiment>_mpi.run`, and `exp.<experiment>_omp.run` to perform simulations A, B, C, D, and E, described in section ??, respectively. The latter scripts are then modified by `icon-dev.checksuite` using `sed` according to the needs of the respective runs.

**diff\_results.** This function compares two simulations. The five arguments contain the base path of the model (`.../icon-dev/` for example), and the name of the experiment to be compared (e.g. `<experiment>_base`) for the two experiments, respectively. The path of the models can be identical (e.g. for the restart or nproma tests that are performed on the same model revision). The fifth argument is the name of the test (`update`, `restart`, `nproma`, `mpi`, `omp`) and is only used to produce more legible output. However, the `diff_results` function needs further information that is provided by variables that are set in the main script: (i) the respective infix of the output files in variable `TYPES` (e.g. `atm_phy`), the output dates and time in variable `DATES` (e.g. `19780101T004000Z`) as they figure on the output filenames, and the restart date in `RESTART_DATE`. These three variables can be set as arguments to the options `-t`, `-d`, and `-s` in a call to `icon-dev.checksuite`, respectively. The `diff_results` function checks for differences between two experiments by the `cdo diff` command. If the variable `SUB_FILES` is set to `'yes'`, e.g. by the use of the `-u` option in the call of `icon-dev.checksuite`, the variables of the respective outputfiles are subtracted from each other resulting in difference files

```
diff_<EXP2>_<TYPE>_<DATE>-<EXP1>_<TYPE>_<DATE>.nc
for <TYPE> in TYPES and <EXP[12]> one of <experiment>_base,
<experiment>_restart, <experiment>_nproma, <experiment>_mpi,
<experiment>_openmp, or <experiment>_update. The difference files are written to
the path of experiment EXP2.
```

### 1.4.3 Usage

There are three different ways to use the “check suite”:

- (i) Start on the command line: The test script `icon-dev.checksuite` can be called on the command line from the `run/checksuite.icon-dev` directory. All the below described options are available on the command line and the full functionality can be used via the command line options easily.
- (ii) Submit to queue: Like buildbot does, it is possible to run `make_runscripts` on a respective test experiment script located in `icon-dev/run` and to submit the resulting run script to the respective queuing system. E.g. from `exp.test_atm_amip`, the runscript `exp.test_atm_amip.run` is generated and can be submitted. `exp.test_atm_amip` is just a link to `checksuite.icon-dev/check.atm_amip`. In order to use the full functionality of `icon-dev.checksuite`, various environment variables have to be set in `exp.test_atm_amip`. This way of calling `run/checksuite.icon-dev` is good for testing on computers with a queuing system. To date, the `exp.test_atm_amip` and `exp.check_oce_omip_160km` are the only test scripts that are available.
- (iii) Buildbot: The script calling `icon-dev.checksuite` can be used by buildbot. In this case, it is

important to check that the correct values of all the environment variables are set in the run scripts mentioned in paragraph (ii).

The calling syntax of `icon-dev.checksuite` is:

```
icon-dev.checksuite [-c] [-d <dates>] [-e <experiment>] [-f yes|no] [-h]
                    [-m b(ase)|u(pdate)|r(estart)|n(proma)|m(pi)|o(mp)
                    |ur|un|um|uo|rn|rm|ro|nm|no|mo|urn|urm|uro|unm|uno|umo
                    |rnm|rno|rmo|nmo|urnm|urno|urmo|unmo|rnmo|urnmo]
                    [-o yes|no ] [-r <reference model path>]
                    [-s <restart_date>] [-t <files>] [-u]
```

Description of options:

- c: colour line output. Colour output should not be used when the script is called by buildbot.
- d: dates for which outputfiles exist. The default depends on the experiment.
- e: experiment on which tests have to be performed. Currently, the non-hydrostatic amip-like experiment `atm_amip_test`, the atmospheric experiment including a grid refinement `atm_icoles_0160km`, the Jablonowski Williamson baroclinic wave test on a hydrostatic atmosphere `atm_jww_hs_test`, and the ocean only experiment `oce_omip_0160km` are supported.
- f: The argument `yes` forces to create run scripts even if they already exist (default), `no` creates run scripts only if they are not yet present.
- h: display help
- m: describes the test mode by its arguments that are one of `b(ase)`, `u(pdate)`, `r(estart)`, `n(proma)`, `m(pi)`, `o(mp)`, `ur`, `un`, `um`, `uo`, `rn`, `rm`, `ro`, `nm`, `no`, `mo`, `urn`, `urm`, `uro`, `unm`, `uno`, `umo`, `rnm`, `rno`, `rmo`, `nmo`, `urnm`, `urno`, `urmo`, `unmo`, `rnmo`, `urnmo`. The first five tests modes describe the sole base run, or the update-, restart-, nproma-, mpi-, and omp-tests, respectively. The last 26 acronyms describe combined tests where each single test is represented by its initial letter. The default test mode is `rnmo`.
- o: The argument of this option can be either `yes` or `no` depending on whether existing test simulations shall be overwritten (`-o yes`) or will be re-used for the current tests (`-o no`). The default is `-o yes`, so all existing experiments are automatically overwritten if not specified otherwise.
- r: The argument of this option gives the absolute path to the reference model. If the test mode includes an update test, it is mandatory. No default.
- s: Restart date for the restart test as given by the time settings in the respective experiment. Default depends on the experiment.
- t: “Types” (infixes) of output files that have to be compared. The infixes depend on the experiment and are set by default accordingly.
- u: If files in the various test runs differ, calculate the difference by `cdo sub`.

Corresponding to the options on the command line, the following environment variables can be set in the `exp.test_<experiment>`:

-c: COLOUR='yes' | 'no'. Not recommended in use with buildbot.  
 -d: DATES=<date\_string>.  
 -e: EXPERIMENT=<name>.  
 -f: FORCE\_MRS='yes' | 'no'  
 -m: MD=<test\_mode>  
 -o: OVERWRITE='yes' | 'no'  
 -r: REFERENCE=<reference\_model\_path>  
 -s: RESTART\_DATE=<restart\_date>  
 -t: TYPES=<file\_type\_infixes>  
 -u: SUB\_FILES='yes' | 'no'

#### 1.4.4 Examples

```
icon-dev.checksuite -o no -c -u -e oce_omip_160km
```

This command runs the `rnmo`, i.e. the restart, `nproma`, `mpi` and `openmp` test on the experiment `oce_omip_160km`. Existing runs are not overwritten (`-o no`), there is colour output (`-c`), and the difference files are calculated between the various test experiments and the base run (`-u`).

```
icon-dev.checksuite -c -f no -m ur -r <path>
```

This command performs the update and restart test (`-m ur`) on the `atm_amip_test` experiment, colour output is switched on (`-c`), the reference model is given in `<path>`, the runscripts are not newly generated if they are already there (`-f no`).

## 1.5 Running the Model (Idealized Cases)

To shed light on the functionality and the quality of the dynamical core, setups for two test cases are presented in the following. Additionally, results of these test cases are shown. These tests are classified in short deterministic test cases (typically a simulation period of about 10-30 days) and tests in a climate mode (typically a multi-year period). This section concentrates on the first class, which starts from prescribed initial conditions (ideally provided in analytic form). The simulation results are either compared to analytic solutions (if available) or high-resolution reference solutions. For a list of available testcases, the reader is referred to the namelist section (??).

### 1.5.1 Jablonowski-Williamson test

The Jablonowski-Williamson Test (?) is a standard test for dynamical cores in global models and can be run for dry dynamics only - as it is intended for- but full physics can be also tested.

Input von Daniel Reinert is expected here.

## Setup

For full physics, two additional namelist parameters are introduced in the `testcase_nml` to control the initial moisture in the atmosphere:

- Here `rh_at_1000hpa` to be set between 0 and 1. The default is set to 0.7 which gives a quite smooth start. If you really want to see early onsets of convection and microphysics you have to tune this parameter.
- `qv_max` is usually set to  $20.e - 3kg/kg$  and refers to the maximum value in the tropics.

## Input Data

GRID

## Results

The **Jablonowski-Williamson steady-state test** is based on a zonally symmetric, strongly baroclinic atmosphere. Initially, it is in a hydrostatic and geostrophic balance and therefore should remain stationary if no perturbation is imposed. Grid irregularities can disturb this stationary conditions and hence the test identifies the presence and magnitude of grid imprinting of a numerical model. For the **Jablonowski-Williamson baroclinic wave test**, a weak (and unbalanced) perturbation disturbs the initial wind. This test highlights the diffusivity (or effective resolution) of a dynamical core and the presence of phase speed errors in the advection of poorly resolved structures.

### 1.5.2 Mountain induced Rossby waves

In order to test the model dynamics in dry stage but with real or any complex topography one can choose the mountain induced Rossby wave test case and select different types of topography. The following namelist parameters give an example how to perform such an idealized simulation.

```
NAMELIST EXAMPLE for moutain induced Rossby waves
! nh_testcase_nml: idealized testcase specification
&nh_testcase_nml
  nh_test_name          = 'mrw_nh' ! testcase selection
  u0_mrw                = 20.0      ! initial u-component
  mount_height_mrw      = 2000.0    ! maximum mountain height
  mount_half_width      = 1.5e06    ! half width of mountain
  mount_longctr_mrw_deg = 90.0      ! longitude: center of the mountain
  mount_latctr_mrw_deg  = 30.0      ! latitude : center of the mountain
/
! run_nml: general switches
&run_nml
  ltestcase = .TRUE. ! idealized testcase runs
```

```

num_lev    = 90      ! number of full levels (atm.) for each domain
lvert_nest = .TRUE.  ! vertical nesting
nsteps     = 1000    ! number of time steps of this run
dttime     = 288     ! timestep in seconds
ldynamics  = .TRUE.  ! compute adiabatic dynamic tendencies
ltransport = .FALSE. ! compute large-scale tracer transport
ntracer    = 0       ! number of advected tracers
iforcing   = 0       ! forcing by parameterized processes
msg_level  = 7       ! controls printout during runtime
ltimer     = .FALSE. ! monitoring the runtime of specific routines
output     = "nml"   ! main switch for components of the model output

```

### Initial conditions

Applying this namelist parameters the topography shown in Fig. ?? is used.

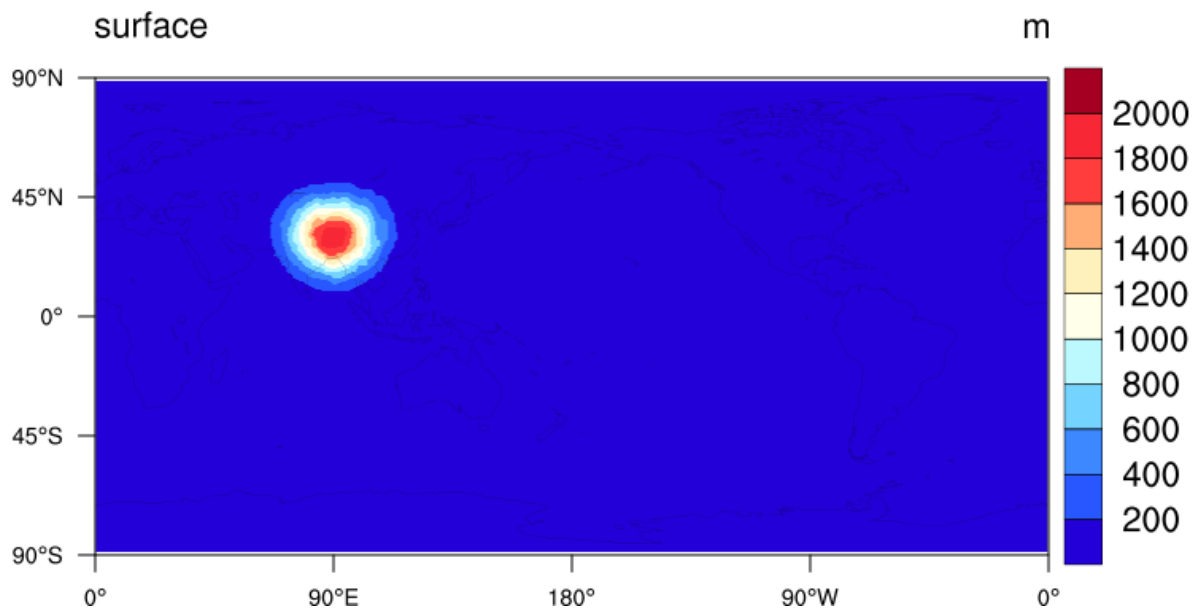


Figure 1.1: Topography of the test case

The v-component of the wind speed is initialized with zero at all grid points, the initial conditions for the u-component are shown in Fig. ??.

### Results after 16 days

The u-component after sixteen days of simulation at 700 hPa is shown in Fig. ??, the corresponding v-component is shown in Fig. ??, and Fig. ?? shows the vorticity.

### Input Data

With the exception of the grid file no further input files are necessary.



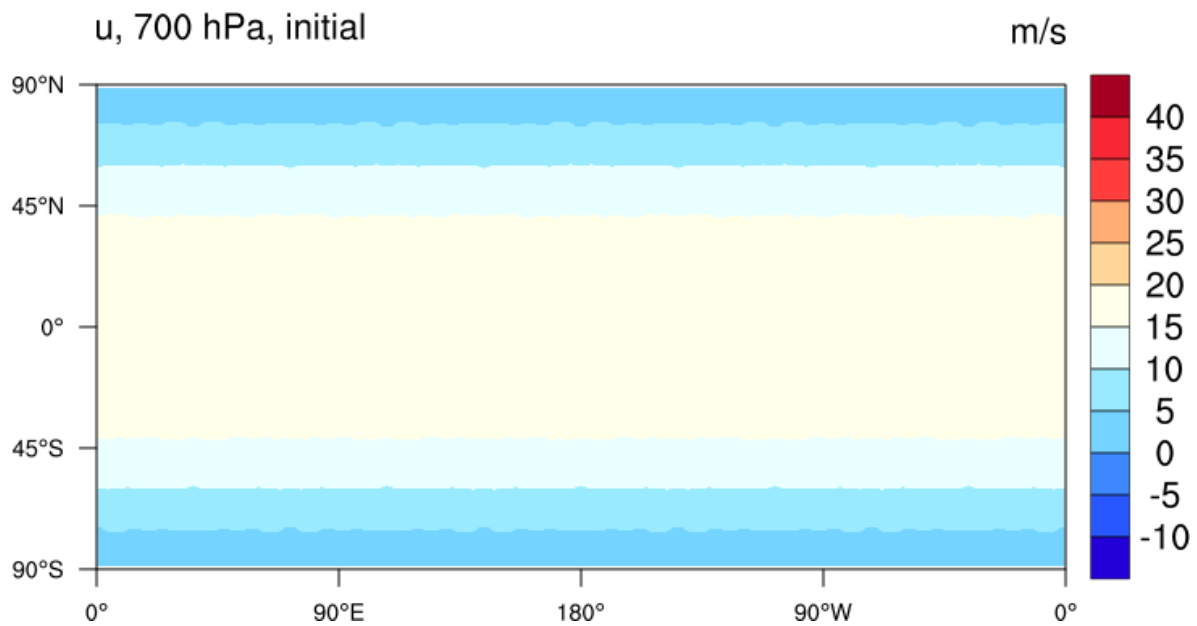


Figure 1.2: Spatial distribution of the initialized u-component at 700 hPa

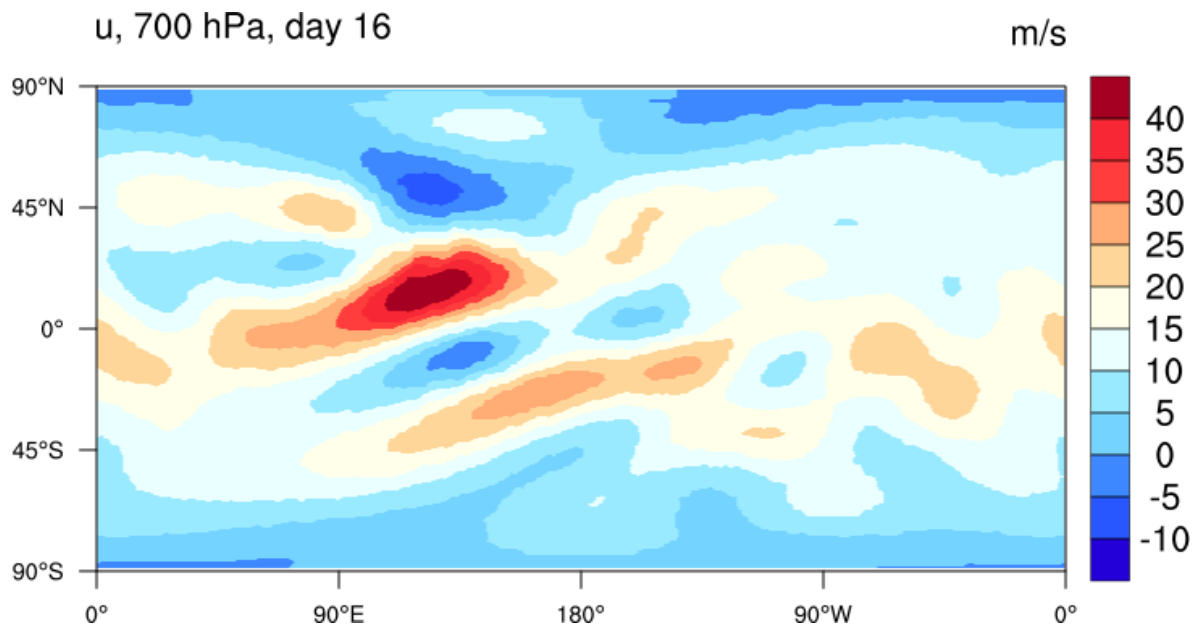


Figure 1.3: Spatial distribution of u-component after 16 days of simulation at 700 hPa

### 1.5.3 cr2016\_02\_16\_rjs: Aqua planet in ICON

The Aqua Planet Experiment (APE) is incorporated in ICON as a set of testcases, all of them having in common that the bottom boundary consists of water only, with all land surface and orography being removed. Originally APEs were constructed as a test-bed for model intercomparison regarding the interaction of dynamics and physical parameterizations in atmospheric General Circulation Models (GCMs) ???. However, this idealized setup is also well suited for studies focusing on the interaction of basic atmospheric phenomena and the ocean surface, e.g. tropical convection, when a realistic representation of the atmosphere is secondary.

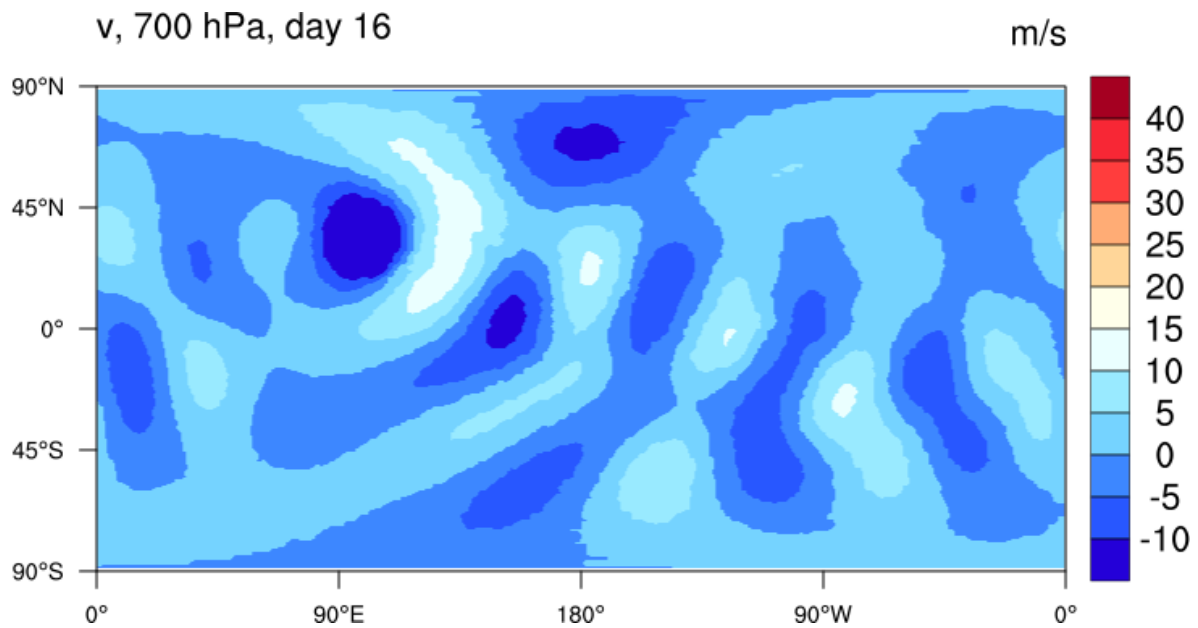


Figure 1.4: Spatial distribution of v-component after 16 days of simulation at 700 hPa

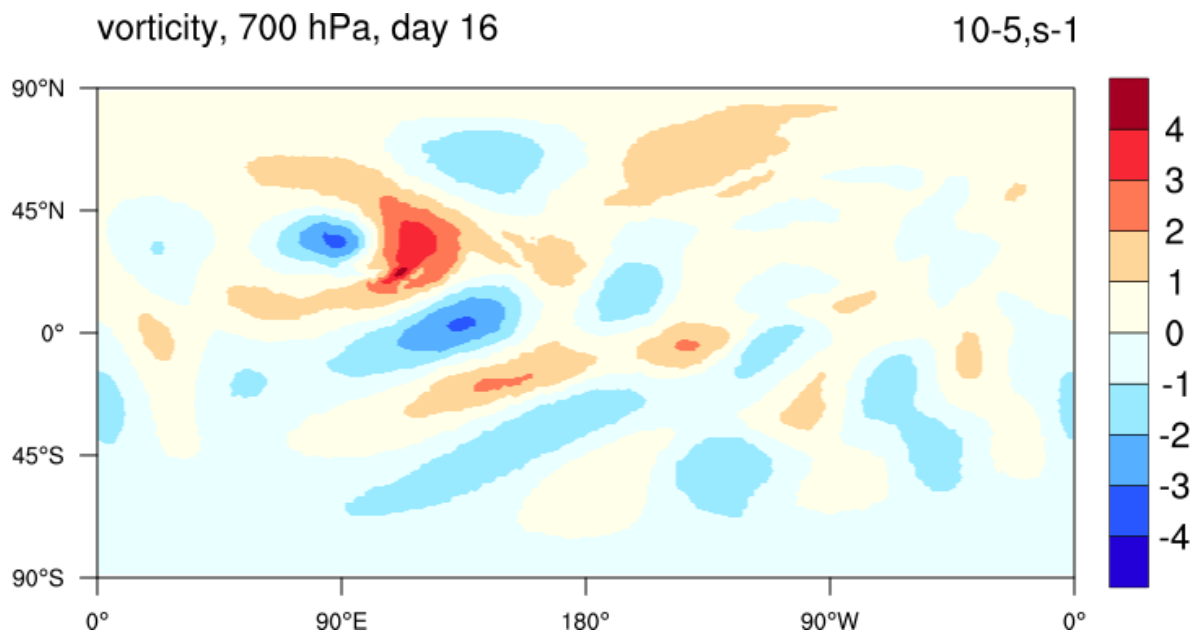


Figure 1.5: Spatial distribution of the vorticity after 16 days of simulation at 700 hPa

The APE is one of the test cases of ICON and can be switched on by setting the respective `nh_test_name` variable in the `nh_testcase_nml` namelist, see Listing ??.

Sea Surface Temperatures (SSTs) can either be prescribed by one of several analytical functions and kept constant in time, or alternatively SSTs may interact with the atmosphere by the use of a slab ocean layer. The slab ocean is not tested yet. The following SSTs  $T_{\text{surf}}$  are available from `mo_ape_params.f90` under the respective keywords ( $T_m = 273.15$  K):

**keyword** `sst1`

$$T_{\text{surf}} = \begin{cases} T_{\text{m}} + 27 \text{ K} \left(1 - (\sin(3\phi/2))^2\right) & \text{for all } -\pi/3 < \phi < \pi/3 \\ T_{\text{m}} & \text{otherwise} \end{cases} \quad (1.1)$$

**keyword** `sst2`

$$T_{\text{surf}} = \begin{cases} T_{\text{m}} + 27 \text{ K} \left(1 - \frac{3}{\pi}|\phi|\right) & \text{for all } -\pi/3 < \phi < \pi/3 \\ T_{\text{m}} & \text{otherwise} \end{cases} \quad (1.2)$$

**keyword** `sst3`

$$T_{\text{surf}} = \begin{cases} T_{\text{m}} + 27 \text{ K} \left(1 - (\sin(3\phi/2))^4\right) & \text{for all } -\pi/3 < \phi < \pi/3 \\ T_{\text{m}} & \text{otherwise} \end{cases} \quad (1.3)$$

**keyword** `sst4`: This distribution has its temperature peak at 5°N:

$$T_{\text{surf}} = \begin{cases} T_{\text{m}} + 27 \text{ K} \left(1 - (\sin(1.6363(\phi - \frac{\pi}{36})))^2\right) & \text{for all } \pi/36 < \phi < \pi/3 \\ T_{\text{m}} + 27 \text{ K} \left(1 - (\sin(1.3846(\phi - \frac{\pi}{36})))^2\right) & \text{for all } -\pi/3 < \phi \leq \pi/36 \\ T_{\text{m}} & \text{otherwise} \end{cases} \quad (1.4)$$

**keyword** `sst_qobs`

$$T_{\text{surf}} = \begin{cases} T_{\text{m}} + \frac{27}{2} \text{ K} \left(2 - (\sin(3\phi/2))^2 \left(1 + (\sin(3\phi/2))^2\right)\right) & \text{for all } -\pi/3 < \phi < \pi/3 \\ T_{\text{m}} & \text{otherwise} \end{cases} \quad (1.5)$$

**keyword** `sst_ice`

$$T_{\text{surf}} = \begin{cases} T_{\text{m}} + 1.9 \text{ K} + 27 \text{ K} \left(1 - (\sin(3\phi/2))^2\right) & \text{for all } -\pi/3 < \phi < \pi/3 \\ T_{\text{m}} - 1.9 \text{ K} & \text{otherwise} \end{cases} \quad (1.6)$$

**keyword** `sst_const`

$$T_{\text{surf}} = T_{\text{m}} + 29 \text{ K} \quad (1.7)$$

The various SSTs can be set in the namelist `nh_testcase_nml` by attributing the corresponding keyword to the variable `ape_sst_case`. For an example see Listing ??.

Listing 1.1: Testcase namelist for APE (`exp.atm_ape_test`)

```
&nh_testcase_nml
  nh_test_name      = 'APE_echam'
  ape_sst_case      = 'sst_qobs'
/
```

The APE can be conducted with the MPI/ECHAM physics only since the use of the PSRAD radiation is essential for the APE. A standard example script `exp.atm_ape_test` is provided. In this standard version, a Kepler orbit with no eccentricity and zero obliquity is used. The orbit parameters are set in the `psrad_orbit_nml` namelist (Listing ??).

Listing 1.2: Orbit namelist for APE (`exp.atm_ape.test`)

```
&psrad_orbit_nml
cecc      = 0.0      ! zero eccentricity
cobld     = 0.0      ! zero obliquity
l_orbvsop87 = .FALSE. ! use Kepler orbit instead of historic one
/
```

If a circular orbit with no obliquity is used, a year with months of a different number of days does not make sense. Instead, a 360 day year should be used. The calendar must be set by two variables at the beginning of the `exp.atm_ape.test` script (Listing ??).

Listing 1.3: Calendar for APE (`exp.atm_ape.test`)

```
calendar=" '360_day_year' "
calendar_type=2
```

The solar irradiation is set to the preindustrial value, i.e. to a value of  $1360.875 \text{ W/m}^2$ . The APE uses a composition of the atmosphere where the  $\text{O}_3$  concentration is read from a special file. Prognostic water vapour and prognostic liquid and ice cloud water is used together with constant  $\text{CO}_2$  in space and time, and no aerosols. All other greenhouse gases are set to zero (Listing ??).

Listing 1.4: Solar irradiation and composition of atmosphere (`exp.atm_ape.test`)

```
&radiation_nml
isolrad   = 2 ! preindustrial solar constant of 1360.875 W/m^ at
            1 AE
irad_h2o   = 1 ! prognostic vapor, liquid and ice
irad_co2   = 2 ! constant co2
irad_o3    = 4 ! perpetual january of ozone file linked
irad_ch4   = 0 ! no ch4
irad_n2o   = 0 ! no n2o
irad_o2    = 0 ! no o2
irad_cfc11 = 0 ! no cfc11
irad_cfc12 = 0 ! no cfc12
irad_aero  = 0 ! no aerosol
/
```

The  $\text{O}_3$  concentrations are read from the file `ape_o3_iconR2B04-ocean_aqua_planet.nc` that is provided in the directory `data/external/ape_ozone` of the ICON main directory. Table ?? lists possible ozone files.

#### 1.5.4 cr2016\_02\_16\_rjs: Radiative–Convective equilibrium in ICON using ECHAM physics

The radiative–convective equilibrium (RCE) offers a possibility to improve our fundamental understanding of processes in the atmosphere and their impact on climate change (e.g. ?). The idea behind this simplified modeling of the atmosphere is that the basic atmospheric structure, especially in the tropics, is determined by the balance between cooling of the atmosphere through radiative processes and a commensurate heating through convection, mainly by the net release of latent heat through precipitation.

Table 1.1: Possible ozone file

file name	description
<code>ape_o3_iconR2B03-ocean_aqua_planet.nc</code>	contains one time step of ozone values, interpolated from <code>ape_o3_T42.1Pa.nc</code> to r2b3 resolution
<code>ape_o3_iconR2B04-ocean_aqua_planet.nc</code>	contains one time step of ozone values, interpolated from <code>ape_o3_T42.1Pa.nc</code> to r2b4 resolution
<code>ape_o3_R2B04_1Pa_spr0.90-cell.nc</code>	contains the same ozone values as the file above
<code>ape_o3_R2B04_hex_1Pa_c.nc</code>	unclear, contains 10242 grid points only instead of 20480 as the files for r2b4 resolution (hexagons instead of triangles?)
<code>tor120000s64_ozone_CMIP5_aqua_1870_march.nc</code>	one time step on 8192 cells, 36 levels, seems to be on a torus grid.

The RCE has been investigated in models of different complexity, ranging from simple energy balance, 1-dimensional column models to high resolution LES simulations. The RCE is also implemented into the general circulation model ICON by creating a model configuration, where the resulting climate is given merely through the balance of radiative processes and convection. Columns can interact with each other and thus create a mean three-dimensional circulation which develops interactively, although it is very different from the general circulation we know from the real Earth. E.g., the RCE results in slowly moving convective clusters of sometimes continental extension (?).

To inhibit net energy transport from the tropics to the poles, homogeneous boundary conditions are specified, where every gridpoint of the sphere receives the same incoming solar radiation (e.g. about  $340 \text{ W/m}^2$ ). A diurnal cycle may be switched on, but is kept exactly the same for each column representing a pulsating light source shining from all directions equally. The Earth's rotation velocity is set to zero. In the standard RCE configuration, land-sea contrasts are removed by specifying an underlying mixed-layer ocean with a constant ocean albedo, but can easily be included in idealized form for land-sea contrast studies (?). This model version has not been tested for possible equilibria dependence on the initial boundary conditions yet, nor for complete isotropy of variables expected from the homogeneous boundary conditions.

### Setting initial and boundary conditions and parameters for the RCE

We describe the settings in the standard example script `exp.atm_rce_test` here. In that case, the mixed layer ocean is switched on by

Listing 1.5: Mixed layer ocean switch in `exp.atm_rce_test`

```
&echam_phy_nml
...
lmlo = .TRUE.
...
/
```

For the initialization of the RCE configuration, we use the `nh_testcase_nml` namelist.

Listing 1.6: Testcase namelist for RCE in `exp.atm_ape_test`

```
&nh_testcase_nml
  nh_test_name      = 'RCE_glb'
  ape_sst_case      = 'sst_const'
  ape_sst_val       = 25.
  tpe_temp          = 298.15
  tpe_psfc          = 1013.25e2
/
```

In namelist `nh_testcase_nml` (Listing ??), the variable `nh_test_name` set to `'RCE_glb'` has the effect to initialize a global model with all values not depending on the geographical position. The primary initialization is done in `init_nh_state_rce_glb` of module `mo_nh_rce_exp.f90`. After this primary initialization, a random noise is added to the variables. The variable `ape_sst_case` is used to initialize the ocean surface temperature by a constant throughout the globe whereas `ape_sst_val` gives the value of the sea surface temperature in degrees centigrade. The variable `tpe_temp` determines the temperature of the atmosphere in Kelvin that is independent of altitude at the beginning. The variable `tpe_psfc` gives the surface pressure in Pascal.

The RCE model can be run in a configuration with the MPI/ECHAM physics only, since the use of the PSRAD radiation is essential for the special radiation settings of the RCE. A uniform irradiation of the globe that may undergo a daily cycle is switched on by setting `l_sph_symm_irr` to `.TRUE.` in namelist `psrad_orbit_nml` (see Listing ??). However, a Kepler orbit with no eccentricity has to be used to warrant a sun-earth distance that is constant in time. This is assured by the three first variables in the namelist `psrad_orbit_nml` as shown in Listing ?. The obliquity is set to zero for convenience.

Listing 1.7: Orbit namelist `psrad_orbit_nml` for RCE in `exp.atm_rce_test`

```
&psrad_orbit_nml
  cecc              = 0.0
  cobld             = 0.0
  l_orbvsop87       = .FALSE.
  l_sph_symm_irr     = .TRUE.
/
```

The angular velocity of the earth is set to zero in order to exclude any quantity that depends on latitude as the Coriolis force. To this end, `grid_angular_velocity` is set to zero in namelist `grid_nml` (see Listing ??).

Listing 1.8: Grid namelist `grid_nml` for RCE in `exp.atm_rce_test`

```
&grid_nml
  grid_angular_velocity = 0.
/
```

The diurnal cycle can be switched on and off with the variable `ldiurn` of namelist `radiation_nml` (see Listing ??).

If a circular orbit with no obliquity is used, a year with months of a different number of days does not make sense. Instead, a 360 day year should be used. The calendar must be set by two variables at the beginning of the `exp.atm_ape_test` script (Listing ??).

Listing 1.9: Calendar for APE (`exp.atm_ape_test`)

```
calendar=" '360_day_year' "
calendar_type=2
```

The solar irradiation has to be set to an average value that corresponds to the actual energy flux into the atmosphere of the real earth. If spherically symmetric irradiation without diurnal cycle is chosen, `isolrad` has to be set to 5, if the diurnal cycle is switched on, `isolrad` has to be set to 4, respectively (see Listing ??). These settings result in a global mean insolation of  $340.3 \text{ W/m}^2$ . However, the sum of the 14 solar wavelength bands is higher ( $433.3371 \text{ W/m}^2$  and  $1069.315 \text{ W/m}^2$  for `isolrad` = 5, 4, respectively), due to the applied solar zenith angles and the eventual diurnal cycle.

The composition of the atmosphere used in the RCE model is described by the corresponding switches in namelist `radiation_nml` (see Listing ??). The  $\text{O}_3$  concentration is read from a special file `r2b4.ozone_rce.nc` that is distributed with the program code in the directory `data/external/ape.ozone` of the ICON main directory. Prognostic water vapour and prognostic liquid and ice cloud water is used together with constant  $\text{CO}_2$  in space and time, and no aerosols. All other greenhouse gases are set to zero (see Listing ??).

Listing 1.10: Radiation namelist `radiation_nml` for RCE in `exp.atm_rce_test`

```
&radiation_nml
  irad_h2o      = 1 ! prognostic vapor, liquid and ice
  irad_co2      = 2 ! constant co2
  irad_ch4      = 0 ! no ch4
  irad_n2o      = 0 ! no n2o
  irad_o3       = 4 ! perpetual january of ozone file linked
  irad_o2       = 0 ! no o2
  irad_cfc11    = 0 ! no cfc11
  irad_cfc12    = 0 ! no cfc12
  irad_aero     = 0 ! no aerosol
  isolrad       = 5 ! 4 if a diurnal cycle is switched on
  ldiur         = .FALSE. ! no diurnal cycle, .TRUE. for diurnal cycle
/
```

## 1.6 Running the Model (Real Case)

The ICON code, as checkout from the SVN repository, does not include runscripts. Instead the run directory (`$ICONDIR/run/`) includes several descriptor files for building grids, defining experiments and post-processings. There exist three different types of descriptor files with prefixes `grid`, `exp`, `post`:

- `grid.<name>`: to configure the grid generator, see chapter ?? for more details. It is recommended to use pre-built grids. For details, see section ??.
- `exp.<name>`: to define the namelist, which determinate the experiments.
- `post.<name>`: to define post-processing.

### 1.6.1 Input Data

Generally ICON requires the following input data: Grid files, external parameters, initialization (DWD analysis or IFS), input fields for radiation.

#### Grid Files

In order to run ICON, it is necessary to have the horizontal grid information as an input parameter. This information is stored within so-called grid files. For a ICON run, one global grid file is necessary. Additionally, if you want to nest, grid files of the nested domains are necessary, too. To improve the performance of ICON, a (optional) reduced radiation grid for each domain may be used.

The naming of the ICON-Grid is as follows: The initial icosahedron grid is refined by  $\langle n \rangle$ -secting the edges, and further refinement is obtained by iteratively bisecting the created edges. The grid produced at the  $\langle k \rangle$ -refining iteration is named "R $\langle n \rangle$ B $\langle k \rangle$ ". For further details, see the ICON Technical Documentation.

It is recommended to use pre-built grids. Further information can be found in chapter ???. For building own grids, the reader is referred to chapter ??. The names of the grid files have to be specified within the `grid_nml`:

```
&grid_nml
dynamics_grid_filename = "<INSERTFILENAME>"
radiation_grid_filename = "<INSERTFILENAME>"
```

#### External Parameters

ICON requires geographical localized datasets like the topographic height of the earth surface, the plant cover, the distribution of land and sea and, dependent on the schemes used, a variety of other so called external parameters. The EXTPAR software system (EXTPAR - External Parameter for Numerical Weather Prediction and Climate Application) is able to generate external parameters for the different models GME, COSMO, HRM and ICON. The software can run on a UNIX or Linux systems where the raw data is stored. It allows operators (experienced users) running the scripts to create new external parameters controlled by user specifications like the model domain. For a more detailed overview of EXTPAR, the reader is referred to the User and Implementation Guide of EXTPAR.

The name of the EXTPAR file which has to be read by ICON can be specified as follows:

```
&extpar_nml
extpar_filename = "<INSERTFILENAME>"
```

If not specified explicitly, ICON uses the following file name:

"<path>extpar\_<gridfile>".

<path> and <gridfile> are then replaced at runtime by ICON.



## Initialization

For the initialization of ICON, input data from either DWD or IFS is needed.

In case of DWD (init\_mode=1) a first guess and an analysis is required:

```
&initicon_nml
dwdfg_filename = "<INSERTFILENAME>"
dwdana_filename = "<INSERTFILENAME>"
```

If not specified explicitly, ICON uses the following file names:

"<path>dwdFG\_R<n>B<k>\_DOM<idom>.nc" and

"<path>dwdana\_R<n>B<k>\_DOM<idom>.nc".

<path>, <n>, <k> and <idom> are then replaced at runtime by ICON according to the chosen gridfile (see ??). The variable <idom> is an index for the domain on which the calculations are performed. <idom>=0000 is reserved for a reduced radiation grid, <idom>=0001 for the global domain, higher numbers are used for nested domains. NETCDF as well as GRIB2 input can be used.

In case of IFS (init\_mode=2) an analysis is required. It has to be in NetCDF:

```
&initicon_nml
ifs2icon_filename = "<INSERTFILENAME>"
```

If not specified explicitly, ICON uses the following file name:

"<path>ifs2icon\_R<n>B<k>\_DOM<idom>.nc".

<path>, <n>, <k> and <idom> are then replaced at runtime by ICON according to the chosen gridfile (see ??). The variable <idom> is an index for the domain on which the calculations are performed. <idom>=0000 is reserved for a reduced radiation grid, <idom>=0001 for the global domain, higher numbers are used for nested domains.

## Radiation

ICON requires input fields for the RRTM radiation scheme. The file names are specified as follows:

```
&nwp_phy_nml
lrrtm_filename = "<INSERTFILENAME>"
cldopt_filename = "<INSERTFILENAME>"
```

If not specified explicitly, ICON uses the following file names:

"rrtmg\_lw.nc" and

"ECHAM6\_CldOptProps.nc".

The files can be found within \$ICONDIR/data.

### 1.6.2 Creating a Runscript

To create a runscript, new users are advised to use the namelist descriptor file **exp.nh-oper** which contains recently recommended namelist settings. It might be necessary to account for the file

names and paths of the input data. Additionally, machine dependent settings need to be added to this script to obtain a runscript. For some architectures, this step can be performed by using the make runscript environment as shown in ???. In the following, example settings for DWD CRAY are listed.

CRAY EXAMPLE: Environment variables

```
#!/bin/ksh
#=====
#PBS -q xc_normal
#PBS -l select=:ncpus=:mpiprocs=:ompthreads=:mem=?gb
#PBS -l place=scatter
#PBS -j oe
#PBS -N <<Jobname>>

export MPICH_RMA_OVER_DMAPP=1
```

CRAY EXAMPLE: Namelists

<<Place your namelists e.g. from exp.nh\_oper here>>

CRAY EXAMPLE: Submitting a job

```
aprun
-n <<INSERT: Total number of MPI Tasks>> \
-N <<INSERT: MPI Tasks/Node>> \
<<INSERT: Hyperthreading e.g. 2 -> 20 physical -> 40 "virtual" cores>> \
-d <<INSERT: Threads/MPI Task>> \
-m <<INSERT: Amount of memory to use>> control_model
```

### 1.6.3 Restart

A restart of the model requires a restart file that has to be created by a previous model run. In the following the procedures and the corresponding namelist settings are explained.

#### Creating the initial restart file:

The first job in a series of model runs creates the first restart file. To do so we have to use the following namelist switches.

```
&master_nml
lrestart = .FALSE.
```

In addition we have to prescribe at which time interval the job should produce a restart file:

```
&io_nml
dt_checkpoint = "<Insert time in seconds>"
```

The ICON run then creates restart files for each domain 1, ..., `n_dom`, and for each restart output time step.

The filenames are generic and look like:

```
"<gridfile>_restart_<modeltype>_<timestamp>.nc",
```

An example would be:

```
"iconR2B06_DOM01_restart_atm_20110101T001200Z.nc"      (NetCDF format)
```

This filename can be customized using the namelist parameter:

```
&mo_run_nml
restart_filename = "<INSERTFILENAME>"
```

This file contains:

- data
- namelists
- several attributes

Note: - ICON reads the namelists only once and assumes that these are identical for all domains.  
 - Since we do not know about the total number of domains at startup, we have to ask the current restart file for the attribute "`n_dom`".

For each domain 1, ..., `n_dom`, a symbolic link is generated with the generic name:

```
"restart_<modeltype>_DOMxx.nc"
```

Note: - The domain-dependent suffix "...DOMxx" is also required for non-nested setups.

### Running the model in the restart mode:

ICON has to be informed that you want to carry out a restart run:

```
&master_nml
lrestart = .TRUE.
```

The generic link "`restart_<modeltype>_DOMxx.nc`" is used by the restart run to point to the last written restart file of the previous model run.

### Chain of restart runs

If a chain of restart runs is foreseen it is recommended to use the namelist parameter `dt_restart`.

```
&time_nml
dt_restart = "<Insert time in seconds>"
```

In this case only one restart file is produced by each model run and after writing the restart file the job stops.

Note:- `dt_restart` and `dt_checkpoint` have to be selected carefully.

### Asynchronous restart:

The restart can be handled by separated processors. The number of restart processors can be chosen by the user. The corresponding namelist parameter is:

```
&parallel_nml
num_restart_procs = n
```

`n` is the number of processors used for restart.

## 1.6.4 Make Runscript Environment

A full listing of descriptor files you will find in `$ICON/run/`.

After configuration and compiling (chapter ??) these descriptor files can be transformed into run-scripts, which should include the necessary system dependent parameters and the execution section `exec.icon` (`$ICONDIR/run/exec.iconrun`), which starts the actual integration. This transformation is done in `$ICONDIR` by:

```
./make_runscripts
```

This transforms every existing descriptor file in `$ICONDIR/run/<type>.<name>` into a ready-to-use run script `$ICONDIR/run/<type>.<name>.run`

For illustration there exists also

```
./make_my_runscripts
```

which transforms a single descriptor file into a run script. This file is an exemplary file and you can see how to define run parameters.

An exemplary descriptor file for a operational run is `exp.nh_oper`.

**Note:** if you change, or create a descriptor you will need to (re)create the run script in order for the changes to take effect.

To run a script `<type>.<name>.run`, either for creating grids or making an experiment or doing post-processing, go to the `./run` folder

```
cd run
```

and use the job submission command, which depends on your machine:

```
[<submit>] <type>.<name>.run
```

[<submit>] is something like: {llsubmit,qsub}

**Note:** Before (!) running an experiment, the ICON grids must be available to the model. For this purpose, either pre-built grids and ExtPar Data can be used (see Sec. ??) or create own grids (??). For a new user, it is suggested to use pre-built grids first.

## 1.7 Pre-built Grids and ExtPar Data

A list of grid files has been pre-built for the ICON model together with the corresponding reduced radiation grids and the external parameters.

1. The **primary storage** location for ICON grids is

```
blizzard:/pool/data/ICON/grids/public
```

2. Every 24h the contents of the primary storage directory are mirrored to DWD's HPC.
3. Every 24h the contents of the primary storage directory are mirrored to a public web site:  
<http://icon-downloads.zmaw.de>.

Each grid file consists of a NetCDF file and a GPG signature file ([http://de.wikipedia.org/wiki/GNU\\_Privacy\\_Guard](http://de.wikipedia.org/wiki/GNU_Privacy_Guard)).

The signature file makes sure that a grid file is complete and verifies the authorship.

### 1.7.1 Grid file nomenclature

The grids are identified by

- a **centre** number
- a **subcentre** number
- a **numberOfGridUsed**  
which is simply an integer number, increased by one with every new 'official' grid.

The grid files and the external parameter files are named accordingly, e.g.,

```
icon_grid_0001_RxxByy_G.nc  
icon_extpar_0001_RxxByy_G.nc
```

where the name components are as follows:

```
icon _ grid _ 0001 _ R 02 B 06 _ R .nc
                        (radiation/reduced)
icon _ extpar _ 0002 _ 03 07 _ G .nc
                        (global)
```

The `numberOfGridUsed` parameter is part of the file name (0001, ...) and makes this file name unique.

In general, a lookup table is required to find the actual file name to which a set of these parameters corresponds. This 'table file' is located under

[http://icon-downloads.zmaw.de/dwd\\_grids.xml](http://icon-downloads.zmaw.de/dwd_grids.xml)

(the table file itself is under version control: [https://svn.zmaw.de/svn/icon\\_grid\\_table](https://svn.zmaw.de/svn/icon_grid_table)).

## 1.8 Grid Generation

### 1.8.1 ICON atmosphere grids

The ICON horizontal spherical grid is based on the projection of the icosahedron on the sphere. This is a 2-dimensional grid, representing the earth's surface. The ICON grids need to be created, stored as NetCDF files, and consequently used by the ICON model. Alternatively, already stored grids may be used.

The initial icosahedron grid is refined by  $\langle n \rangle$ -secting the edges, and further refinement is obtained by iteratively bisecting the created edges. The grid produced at the  $\langle k \rangle$ -refining iteration is named "R $\langle n \rangle$ B $\langle k \rangle$ ", and the corresponding NetCDF-file is "iconR $\langle n \rangle$ B $\langle k \rangle$ -grid.nc". The grid files, after their creation, are located in the `./grids` folder. For more detailed information about horizontal ICON grids the reader is referred to the ICON technical documentation.

Examples of grids are in `./grids`. More information can be found in:  
`$ICONDIR/doc/technical/icon_grid.pdf`.

The example given below shows the namelist parameters for generating a global R2B6 grid.

```
EXAMPLE Grid Generation of a R2B6 grid
#!/bin/ksh
#-----
# Creation of atmosphere grids for ICON
#-----
# ICON grid generator namelist parameters
#
# For a complete list see Namelist_overview and Namelist_overview.pdf
#
# nroot      = Number of sections into which the edges of the original
#              icosahedron are divided in the initial refinement step.
```

```

#           (icosahedron = "grid -1" --> "nroot" grid = grid "0")
#
# grid_levels = Number of refinement steps applying edge bisection,
#               follows the initial "nroot" refinement step.
#               (grid "0" --> grid "1" --> ... --> grid "grid_levels")
#
# itype_optimize grid optimization method applied from grid level 1 onward.
#               i | optimization      | suffix fo grid output file
#               -----
#               0 | none              | noo
#               1 | Heikes Randall    | hro
#               4 | spring dynamics    | spr
#
# beta_spring = Tuning parameter for spring dynamics to be chosen in the
#               range [0.9,1.1]. Weights the target length between the
#               grid points.
#
#-----
# First generate graphs
R=2      # nroot (the first dissection will be a bisection)
B=6      # highest grid level to reach (number of consequent bisections)
maxlev_optim=6 # highest grid level to apply optimizations

cat > NAMELIST_GRAPH << EOF
&graph_ini
  nroot      = ${R}
  grid_levels = ${B}
/
EOF

echo global_graph_generator null > $commandFile
${start} ${run_commmmand}
check_error $? "global_graph_generator"

#-----
# Generate grids using the spring dynamics optimization
cat > NAMELIST_GRID << EOF
&grid_ini
  nroot      = ${R}
  grid_levels = ${B}
/
&grid_options
  itype_optimize = 4      ! 1 = Heikes-Randall, 4 = spring dynamics
  maxlev_optim = $maxlev_optim ! the maximum level to optimize
/
EOF

```

ICON gives the possibility to nest subdomain within a parent grid. The example below gives the namelist parameters for generating a nested grid (patch). The root bisection of the patch in this

example starts with the fourth level of the bisections of the global model.

```
EXAMPLE Nested grid based on the global grid described before.
#-----
# Next the patches will be created.
# If the pathes are not needed then uncomment the next exit command
#
#exit
#-----
#
# ICON prepare_gridref namelist
#
# Parameter overview:
#
# grid_root: Number of root bisections
#
# start_lev: Grid level of global domain
#
# n_dom:      Total number of model domains (including the global one)
#
# parent_id: List of parent domain ID's (starts at first nested domain,
#           which has ID=2)
#
# l_circ:     true = circular subdomains, false =
#           rectangular (lat/lon) subdomains
#
# l_rotate:   true: rotate center point into equator in case of l_circ=false
#           this yields truly rectangular subdomains, whereas subdomains
#           are conical otherwise because of the convergence of meridians
#
# NOTE:       For subdomains crossing a pole, either l_circ=true
#           or l_rotate=true is required
#
# l_plot:     true: Generates GMT files for domain configuration
#
# NOTE:       The following parameters have to be specified for each nested
#           domain!
#
# radius:     radius (deg) of nested domains (for l_circ=true)
#
# center_lon: Center longitude of nested domains
#
# center_lat: Center latitude of nested domains
#
# hwidth_lon: half-width longitude of nested domains (for l_circ=false)
#
# hwidth_lat: half-width latitude of nested domains (for l_circ=false)
#
#-----
```



```
# suffix of grid files which specifies optimization type
# (without optimization leave empty)
#OPTFIX=spr0.90_M4
# NOTE: _M4 means that maxlev_optim = 4 has to be set in the grid generator
# maxlev_optim is not needed for Heikes-Randall optimization
#
#-----
# Create plots of domain configuration
PLOTS=.false.
#
cat > NAMELIST_GRIDREF << EOF
&gridref_ini
  grid_root   = 2
  start_lev   = 4
  n_dom       = 2
  parent_id   = 1,
  l_circ      = .true.
  l_rotate    = .true.
  l_plot      = .true.
  radius      = 30.,
  center_lon  = -90.,
  center_lat  = 40.,
  hwidth_lon  = 55.,
  hwidth_lat  = 55.,
  bdy_indexing_depth = 14
/
EOF
```

### 1.8.2 Information contained in grid files

The ICON grids are treated as a general unstructured grid, so the grid NetCDF-files contain the full information of the location and the connectivity of all the grid entities (cells, edges and vertices). The grid nesting hierarchy information is also included.

Some basic variables that may be useful for plotting are:

```
double clon(cell)           : longitude of cell centers [radian]
double clat(cell)           : latitude of cell centers [radian]
double clon_vertices(cell, nv) : longitudes of the vertices of the cell [radian]
double clat_vertices(cell, nv) : latitudes of the vertices of the cell [radian]
double elon(edge)           : longitude of edge midpoint [radian]
double elat(edge)           : latitude of edge midpoint [radian]
double elon_vertices(edge, no) : longitudes of the vertices of the edges [radian]
double elat_vertices(edge, no) : latitudes of the vertices of the edges [radian]
double vlon(vertex)         : longitude of vertices [radian]
double vlat(vertex)         : latitude of vertices [radian]
...
double cell_area(cell)      : area of grid cell [m2]
double cell_elevation(cell) : elevation at the cell centers [m]
int cell_sea_land_mask(cell): sea (-2 inner, -1 boundary)
```

```

...
                                land (2 inner, 1 boundary) mask for the cell
...
double edge_length(edge)         : lengths of edges of triangular cells [m]
double dual_edge_length(edge)    : lengths of dual edges (distances between
                                triangular cell circumcenters) [m]
...

```

For a full listing of variables contained in a grid file, for instance in iconR2B04-grid.nc, use:

```
ncdump -h iconR2B04-grid.nc
```

or

```
cdo sinfov iconR2B04-grid.nc
```

### 1.8.3 Viewing/plotting grids

In order to plot an icon grid you should ensure that `ncl-6.0` and `cdo-1.5.4` is available on your machine. Then go to the `$ICONDIR/grids/` folder and give:

```

alias iplot="ncl $ICONDIR/scripts/postprocessing/tools/icon_plot.ncl
'altLibDir="$ICONDIR/scripts/postprocessing/tools/"'" iplot 'iFile="<grid file name>"'
'mapType="ortho"' 'varName="cell_sea_land_mask"' 'oType="png"' 'showGrid=True'
'lStrg="Cell sea land mask"' 'bStrg=""'

```

The above example will plot cell sea land mask. More details on plotting can be found at the Visualization chapter.

The `$ICONDIR/run/post.plot_icon_grids` script can be used to plot nested grids. Go to `$ICONDIR/run/` folder and give:

```
./post.plot_icon_grids
```

A PDF-file with a plot of the iconR2B04\_DOM01 and iconR2B05\_DOM02 grids will appear on your screen. (Note that this process is time consuming.)

**Discussion**

Document last edited by *B Vogel* on *27-06-2014*.

## 2 Output

In general the user has to specify six individual quantities to generate output of the model. These are:

1. The time interval between two model outputs.
2. The name of the output file.
3. The name of the variable.
4. The type of the vertical output grid (e.g. pressure levels or model levels).
5. The type of the horizontal output grid (e.g. ICON grid or geographical coordinates).

ICON offers the possibility to write groups of variables. In the following we will present two examples to demonstrate the options the user has to prescribe these quantities. A detailed description of all namelist parameters available to organize the output is described in `io_nml` in the namelist section.

### Example 1

We will begin with an individual variable which is written in NETCDF format on pressure levels and is interpolated to a horizontally regular lat-long grid:

```
NAMELIST EXAMPLE
&io_nml
  filetype           = 4    ! output format: 2=GRIB2, 4=NETCDFv2
  dom                = 1    ! write output for domain 1
  output_bounds      = 0., 1.E7, 3600. ! start, end, interval in s.
  steps_per_file     = 50   ! max. num. of time steps within one file
  mode               = 1    ! 1: forecast mode (relative t-axis)
  include_last       = .TRUE. ! include the last time step
  output_filename    = '<INSERTFILENAME>' ! file name base
  pl_varlist         = 'geopot' ! name of pressure level field
  remap              = 1    ! output is transferred to lat long grid
  reg_lon_def        = 0.,0.5,359.5 !start, incr., end, in deg.
  reg_lat_def        = 90.,-0.5, -90. !start, incr., end, in deg.
```

### Example 2

The flexibility of the options ICON offers is demonstrated in another example. Now we apply an alternative to define the runtime of ICON, write several variables, at the same time, in one data

set, on model levels, and on the original horizontal grid of ICON. In addition the example below shows the options when several model domains run at the same time and we want to produce output for all model domains.

```
NAMELIST EXAMPLE
&output_nml
  dom                      = -1 ! write all domains
  steps_per_file           = 5  ! max. num. of time steps within
  output_start             = "1978-01-01T00:00:00Z" ! ISO-format date+time
  output_end               = "1979-01-02T00:00:00Z" ! ISO-format date+time
  output_interval          = "PT01H"                ! ISO-format interval
  file_interval            = "PT01D"                ! ISO-format interval
  include_last             = .FALSE.
  output_filename          = '<INSERTFILENAME>'      ! file name base
  ml_varlist='u', 'group:precip_vars' ! Indiv. variable and variable group
  output_grid              = .TRUE. ! Output on the ICON horizontal grid
```

## Variable groups

Next we explain the meaning of variable groups. Using the "group:" keyword for the namelist parameters `ml_varlist`, `hl_varlist`, `pl_varlist`, sets of common variables can be added to the output.

There exists a special syntax which allows to remove variables from the output list, e.g. if these undesired variables were contained in a previously selected group.

Typing "`-<varname>`" (for example "`-temp`") removes the variable from the union set of group variables and other selected variables. Note that typos are not detected but that the corresponding variable is simply not removed!

## How to find variable names and contents of variable groups

Finding the correct names of the variables you may want to write to a data set is not an easy task and you should be aware of some pitfalls. We will help you to avoid the most obvious ones. First of all users that have already experience with the COSMO model should know that the names of the atmospheric variables in ICON are **not identical**.

The easiest way to identify the correct names of the variables you would like to write is to look into the following data sets:

```
atm_dyn_iconam/mo_nonhydro_state.f90
atm_phy_nwp/mo_nwp_phy_state.f90
lnd_phy_nwp/mo_nwp_lnd_state.f90
```

Now you may want to use the option of writing groups of variables and of course you may want to know which variable belongs to which group. Keep in mind that there is an option mentioned before to remove variables from the output of a group of variables.

The following table gives overview on the allocation of variables to individual variable groups. If

you want to translate the Fortran variables to the physical or mathematical ones again have a look to the Fortran files listed above.

```
*****
      nh_prog_vars
vn
rho
theta_v
exner
*****
      dwd_fg_atm_vars
vn
w
rho
theta_v
tke
u
v
pres_sfc
temp
pres
z_ifc
t_2m
td_2m
u_10m
v_10m
*****
      mode_dwd_fg_in
vn
w
rho
theta_v
tke
t_g
t_mnw_lk
t_wml_lk
h_ml_lk
t_bot_lk
c_t_lk
t_b1_lk
h_b1_lk
qv_s
w_i
w_so_ice
w_snow
rho_snow
t_snow_mult
rho_snow_mult
wliq_snow
wtot_snow
```

```

dzh_snow
gz0
*****
        atmo_ml_vars
w
tke
u
v
temp
pres
*****
        atmo_pl_vars
w
tke
u
v
temp
*****
        atmo_z1_vars
w
tke
u
v
temp
pres
*****
        mode_dwd_ana_in
u
v
temp
pres
t_ice
h_ice
fr_seaice
w_so
t_snow
h_snow
freshsnow
*****
        atmo_derived_vars
omega
div
vor
*****
        land_vars
t_g
qv_s
w_i
w_p

```

```

w_s
t_so
w_so
w_so_ice
t_snow
w_snow
rho_snow
snowfrac
*****
            dwd_fg_sfc_vars
t_g
t_ice
h_ice
fr_seaice
w_i
t_so
w_so
w_so_ice
t_snow
w_snow
rho_snow
h_snow
freshsnow
t_snow_mult
rho_snow_mult
wliq_snow
wtot_snow
dzh_snow
gz0
*****
            mode_combined_in
t_g
t_ice
h_ice
qv_s
fr_seaice
w_i
w_so
t_snow
w_snow
rho_snow
h_snow
freshsnow
*****
            mode_cosmode_in
t_g
t_ice
h_ice
qv_s

```



```

w_i
w_so
t_snow
w_snow
rho_snow
h_snow
freshsnow
*****
            dwd_fg_scf_vars
t_mnw_lk
t_wml_lk
h_ml_lk
t_bot_lk
c_t_lk
t_b1_lk
h_b1_lk
qv_s
*****
            land_tile_vars
t_g_t
t_s_t
w_i_t
w_p_t
w_s_t
t_so_t
w_so_t
w_so_ice_t
t_snow_t
w_snow_t
rho_snow_t
t_snow_mult_t
wtot_snow_t
wliq_snow_t
rho_snow_mult_t
dzh_snow_t
qv_s_t
h_snow_t
snowfrac_t
snowfrac_lc_t
*****
            snow_vars
t_snow
rho_snow
wliq_snow
wtot_snow
dzh_snow
*****
            multisnow_vars
t_snow_mult

```

```

rho_snow_mult
wliq_snow
wtot_snow
dzh_snow
*****
    precip_vars

rain_gsp
snow_gsp
rain_con
snow_con
ice_gsp
graupel_gsp
hail_gsp
tot_prec
*****
    additional_precip_vars
con_prec_rate_avg
gsp_prec_rate_avg
cape
clct
tot_cld_vi
*****
    pbl_vars
gust
shfl_s
lhfl_s
lhfl_bs
lhfl_pl
ghfl_s
tcm
tch
t_2m
qv_2m
td_2m
u_10m
v_10m
tkvm
tkvh
*****
    cloud_diag
clc
gc_dia
gi_dia
tot_cld
*****
    rad_vars
thb_s
sod_t

```

```

sou_t
sod_s
sou_s
thd_s
thu_s
sodird_s
sodifd_s
sodufu_s
albdif
albvisdiff
albnirdiff
sob_s_t
thb_s_t
flxdswswtoa
sob_s
sob_t
*****
        phys_tendencies
ddt_temp_radsw
ddt_temp_radlw
ddt_temp_turb
ddt_temp_drag
ddt_u_turb
ddt_u_sso
ddt_u_gwd
ddt_v_turb
ddt_v_sso
ddt_v_gwd:
*****
        prog_timemean
temp_m
rho_m
u_m
v_m
pres_sfc_m
pres_msl_m
*****
        echam_timemean
cosmu0_m
flxdswswtoa_m
aclcov_m
rsfl_m
rsfc_m
ssfl_m
ssfc_m
totprec_m
qvi_m
xlvi_m
xivi_m

```

```

swflxsfc_m
swflxtoa_m
lwflxsfc_m
lwflxtoa_m
tsfc_m
evap_m
lhflx_m
shflx_m
u_stress_m
v_stress_m
*****
      tracer_timemean
qc_m
qv_m
qi_m
*****
      atmo_timemean
all vars of prog_timemean, echam_timmean, tracer_timemean

```

## Data format

ICON offers the possibility to produce output either in NETCDF or GRIB2 format. This can be chosen by the namelist parameter `filetype` of the namelist `&output_nml`. New users are suggested to set `filetype=4` in order to use NETCDF output.

In GRIB2, a variable is uniquely defined by the following set of metadata:

- *Discipline* (see GRIB2 code table 4.2)
- *ParameterCategory* (see GRIB2 code table 4.2)
- *ParameterNumber* (see GRIB2 code table 4.2)
- *typeOfFirstFixedSurface* and *typeOfSecondFixedSurface* (see GRIB2 code table 4.5)
- *stepType* (instant, accum, avg, max, min, diff, rms, sd, cov, ...)

A documentation of the official WMO GRIB2 code tables can be found on the website of WMO: [http://www.wmo.int/pages/prog/www/WMOCodes/WMO306\\_v12/LatestVERSION/WMO306\\_v12\\_GRIB2\\_CodeFlag-en.pdf](http://www.wmo.int/pages/prog/www/WMOCodes/WMO306_v12/LatestVERSION/WMO306_v12_GRIB2_CodeFlag-en.pdf).

## Time stamp format

The namelist parameters `output_start`, `output_end`, `output_interval` allow the specification of time stamps according to ISO 8601. The general format for time stamps is `YYYY-MM-DDThh:mm:ss` where `Y`: year, `M`: month, `D`: day for dates, and `hh`: hour, `mm`: minute, `ss`: second for time strings. The general format for durations is `PnYnMnDTnHnMnS`. See, for example, [http://en.wikipedia.org/wiki/ISO\\_8601](http://en.wikipedia.org/wiki/ISO_8601) for details and further specifications.

NOTE: as the mtime library underlying the output driver currently has some restrictions concerning the specification of durations:

1. Any number  $n$  in  $PnYnMnDTnHnMnS$  must have two digits. For instance use "PT06H" instead of "PT6H"
2. In a duration string  $PnyearYnmonMndayDTnhrHnminMnsecS$  the numbers  $nxyz$  must not pass the carry over number to the next larger time unit:  $0 \leq nmon \leq 12$ ,  $0 \leq nhr \leq 23$ ,  $0 \leq nmin \leq 59$ ,  $0 \leq nsec \leq 59.999$ . For instance use "PT01D" instead of "PT24H", or "PT01M" instead of "PT60S".

Soon the formatting problem will be resolved and the valid number ranges will be enlarged. (2013-12-16).

### Extra output

1. In the namelist `run_ctl` set the number of fields with `inextra_2d` or `inextra_3d`. The logical variable for output `lwrite_extra` then will be set automatically. Note, the number of extra fields is limited by 9 each for 2D and 3D.
2. USE these variables in the module needed.
3. Implement the storage of wished fields by using the nonhydrostatic diagnostic type with `p_diag%extra_2d/3d`.

Example for the use of `p_diag%extra_2d`:

```
USE mo_global_variables, ONLY: inextra_2d
...
DO jc = i_startidx, i_endidx
  p_diag%extra_2d(jc,jb,1) = yxz(jc,jb)
ENDDO
```

### Asynchronous output:

It is highly recommended that the asynchronous output option of ICON is applied. In short this option reserves a number of processors for output only. While writing the remaining processors continuously carry out calculations. Otherwise they would have to wait until output is finished. The corresponding namelist parameter is:

```
&parallel_nml
num_io_procs = n
```

$n$  is the number of processors used for output.

### Time mean output:

The builtin functionality for getting time-averaged output fields preliminary is in an intermediate state. It has the following limitation:

- The list of variables for which time averages can be obtained is fixed. There is no way to change this via namelist
- Respective variables are collected into groups: `prog_timemean`, `echam_timemean`, `tracer_timemean` (all: `atmo_timemean`)
- When time-averaged output is selected, only a single output interval per model component is allowed for the whole experiment
- Output interval has to be a divisor of the restart interval, because the intermediate accumulation results are not saved
- The output for the initial timestep of the time mean variables is zero.

**Testing Time mean:** There are 3 tests prepared for checking the correctness all based on the amip setup: `exp.atm_amip_acc_dtime`, `exp.atm_amip_acc_2dtime` `exp.atm_amip_acc_dtime_sp`, which are all located in `run/checksuite.icon-dev/timeMean`. All of them have a runtime of 80 minutes with timestep `dtime = 20min`:

- `exp.atm_amip_acc_dtime_sp`: This test has output for time mean variables and their instantaneous counterparts versions each time step. It uses the default single precision for output.
- `exp.atm_amip_acc_dtime`: Same as above, but with double precision output
- `exp.atm_amip_acc_2dtime`: Same as double precision, but with output every second timestep

To get binary identical data for online and offline computed mean values, double precision *has* to be used for output and for IO-related MPI communication. In order get the latter activated the variable `use_dp_mpi2io` from the `parallel_nml` has to be set to `TRUE`.

The testing is splitted into 2 steps. The first should ensure, that the output of the instantaneous variables is identical to the output if the same variables averaged over one timestep. This can be done with `exp.atm_amip_acc_dtime_sp`. The values of the variable pairs should be identical in the output file: The shell script `exp.check_timemean.01` can be used for this purpose.

<b>time mean</b>	<code>cosmu0_m</code>	<code>fldwswwtoa_m</code>	<code>acfcov_m</code>	<code>rsfl_m</code>	<code>rsfc_m</code>	<code>ssfl_m</code>	<code>ssfc_m</code>	<code>totprec_m</code>	<code>qvi_m</code>	<code>xlvi_m</code>	<code>xivi_m</code>
<b>instant</b>	<code>cosmu0</code>	<code>rsdt</code>	<code>clt</code>	<code>prlr</code>	<code>prcr</code>	<code>prls</code>	<code>prcs</code>	<code>pr</code>	<code>prw</code>	<code>cllvi</code>	<code>clivi</code>
<b>time mean</b>	<code>swflxsfc_m</code>	<code>swflxtoa_m</code>	<code>lwflxsfc_m</code>	<code>lwflxtoa_m</code>	<code>tsfc_m</code>	<code>evap_m</code>	<code>lhflx_m</code>	<code>shflx_m</code>	<code>u_stress_m</code>	<code>v_stress_m</code>	
<b>instant</b>	<code>rsns</code>	<code>rsnt</code>	<code>rlns</code>	<code>rlnt</code>	<code>ts</code>	<code>evspsbl</code>	<code>hfls</code>	<code>hfss</code>	<code>tauu</code>	<code>tauv</code>	
<b>time mean</b>	<code>xlvi_m</code>	<code>xivi_m</code>	<code>swflxsfc_m</code>	<code>swflxtoa_m</code>	<code>lwflxsfc_m</code>	<code>lwflxtoa_m</code>	<code>tsfc_m</code>	<code>evap_m</code>	<code>lhflx_m</code>	<code>shflx_m</code>	
<b>instant</b>	<code>cllvi</code>	<code>clivi</code>	<code>rsns</code>	<code>rsnt</code>	<code>rlns</code>	<code>rlnt</code>	<code>ts</code>	<code>evspsbl</code>	<code>hfls</code>	<code>hfss</code>	
<b>time mean</b>	<code>qc_m</code>	<code>qv_m</code>	<code>qi_m</code>								
<b>instant</b>	<code>clw</code>	<code>hus</code>	<code>cli</code>								
<b>time mean</b>	<code>u_m</code>	<code>v_m</code>	<code>temp_m</code>	<code>pres_sfc_m</code>	<code>pres_msl_m</code>	<code>z_mc_m</code>	<code>rho_m</code>				
<b>instant</b>	<code>ua</code>	<code>va</code>	<code>ta</code>	<code>ps</code>	<code>psl</code>	<code>zg</code>	<code>rho</code>				

The second check compares online and offline computations. Therefore both `exp.atm_amip_acc_dtime` and `exp.atm_amip_acc_2dtime` has to be run. For all variables in the **time mean** group, the values of the `exp.atm_amip_acc_2dtime` experiment have to be equal to the mean values of the corresponding variables and timesteps in the results of `exp.atm_amip_acc_dtime`.

For example the time mean of timesteps 2 and 3 from `exp.atm_amip_acc_dtime` should have no difference to timestep 2 of `exp.atm_amip_acc_2dtime`. `exp.check_timemean.02` executes this check.

**Example Usage** For getting 6-hourly averaged output, two namelist variable of `output_nml` has to be set up:

- `output_interval = "PT06H"`
- `ml_varlist` should contain the desired list of variables e.g. `'u_m', 'v_m', 'qv_m', 'rho_m', 'temp_m', 'xlvi_m', 'cosmu0_m'` or `'group:prog_timemean', 'group:echam_timemean'`

## Discussion

Document last edited by *I. Kraut* on *29.11.2013*

Document last edited by *S Gruber* on *08-01-2014*

Document last edited by *B Vogel* on *27-05-2014*

Document last edited by *B Vogel* on *30-06-2014*

Document last edited by *R. Müller* on *26-03-2015*

## 3 Visualization

Visualizing data on a non-regular grids is a task on its own, because the number of tools for solving such problem is very limited. NCL is one of them and we chose it as the main tool for ICON. You can find several examples of how to write simple plot scripts for ICON data sets on this website: <http://www.ncl.ucar.edu/Applications/icon.shtml>. The coordinate information is essential for writing your own plot scripts. ICON output files currently have three different types of them: cells, edges and vertices, e.g. tracers like temperature and salinity and surface elevation are defined on each cell center while the normal velocity is defined on edges.

### 3.1 icon\_plot.ncl

For getting around the different coordinates and in order not to rewrite things there is a general plot scripts: `icon_plot.ncl`. It supports contour and vector plots, a combination of both via overlaying and vertical sections. Both atmosphere and ocean vertical coordinate systems can be handled by it: While ocean uses a plain depth axes, atmosphere model uses hybrid sigma pressure levels (hydrostatic) and free 3D height variable (non-hydrostatic).

The script `icon_plot.ncl` is a single NCL program, which provides multiple plot types for data on ICON's grid. It is located in the ICON-repository under `source:/trunk/icon-dev/scripts/postprocessing/tools/icon_plot.ncl`. Most of the functionality is implemented in a library: `icon_plot_lib.ncl` located in `source:/trunk/icon-dev/scripts/postprocessing/tools/icon_plot_lib.ncl`. Both files are installed into the `/pool/data/ICON/tools` which is the default lookup location for the library. For different location like an icon checkout, use `altLibDir`, e.g.

```
altLibDir="/home/user/src/icon-dev/tools".
```

#### 3.1.1 Requirements

- NCL 5.2.1 is the minimum version of NCAR's plotting language (<http://www.ncl.ucar.edu>)
- CDO (<https://code.zmaw.de/projects/cdo>)

#### 3.1.2 Customization

`icon_plot.ncl` optionally reads a configuration file named `$HOME/.icon_plot.rc` where default options can be set. Actually it is handled like an ordinary ncl file. This can be used to customize the `altLibDir` setting, e.g.:

```
altLibDir="/home/ram/src/git/icon/scripts/postprocessing/tools"
oType="png"
```



### 3.1.3 Basic command line option

Required are options for

1. **Input/output files:** Use the variable `iFile` for defining the input and `oFile` for the output file. It's extension depends on the output type, which can be set with `oType`. If `oFile` is left out, the output file will inherit its name from the input file.
2. **Variable selection:** Depending on the plot mode you like to use, `varName` for scalar variables or `vecVars` for vector-variables must be used.

Optional (default:0) parameter are

1. **Level selection:** Levels can only be selected by their index. That's why, the corresponding variable is called `levIndex`. Please note that it starts with 0, like any other NCL indices.
2. **Time selection:** Like `levIndex`, the variable `timeStep` can be used to select a certain time step, again starting from 0.

There are many more parameters (see ??) for mapping, transections, selecting regions and masking, but these are the most fundamental ones.

### 3.1.4 Plot Types

For flexibility the selection of a specific plot mode is implemented by combining certain options.

#### Contour plots

Contour plot are the default plot mode. If only the required parameters are set, e.g. `iFile` and `varName`, a simple contour plot is created with

```
ncl icon_plot.ncl 'varName="T"' 'iFile="iFILENAME"'
```

This is a basic temperature plot. Captions are set to basic information like variable name, time and level information and input filename.

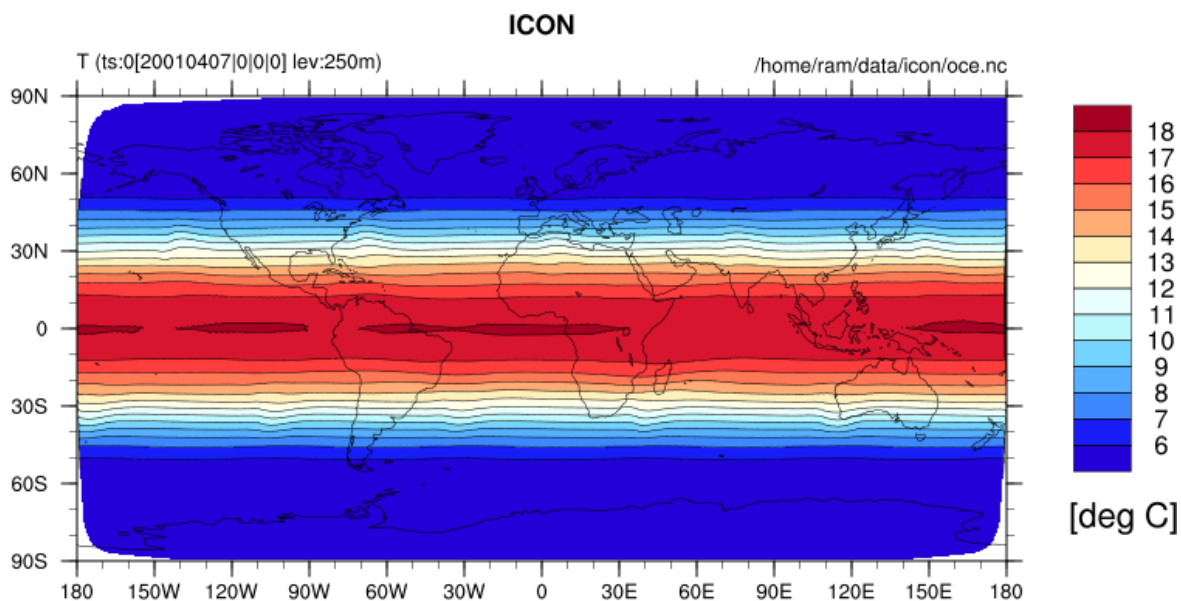
#### Vector plots

Use `vecVars` instead of `varName`. To adjust the length of the reference vector, use the variable `vecRefLength`.

```
ncl icon_plot.ncl 'vecVars="U V"' 'iFile="iFILENAME"' vecRefLength=0.01
```

#### Overlay of scalar and vector variables

Contour and vector plots can be combined into a single plot by overlaying both. Following this approach, such an overlay plot will be created, if `varName` and `vecVars` are given:



Prgr icon\_plot.ncl: Wed Nov 30 14:28:49 CET 2011,ram

Figure 3.1: Example of contour plot

```
ncl icon_plot.ncl 'varName="T"' 'iFile="iFILENAME"' 'vecVars="U V"'
```

### Vertical sections

Data for sections have to be interpolated first. This is done internally and you do not have to care about it. Section plots are created, if a start and end point of a section is given. For this purpose, the variables `secLC` (section-left-corner) and `secRC` (section-right-corner) have to be used. These variables have to be (lon,lat) arrays like `secLC=(/20.,30./)`.

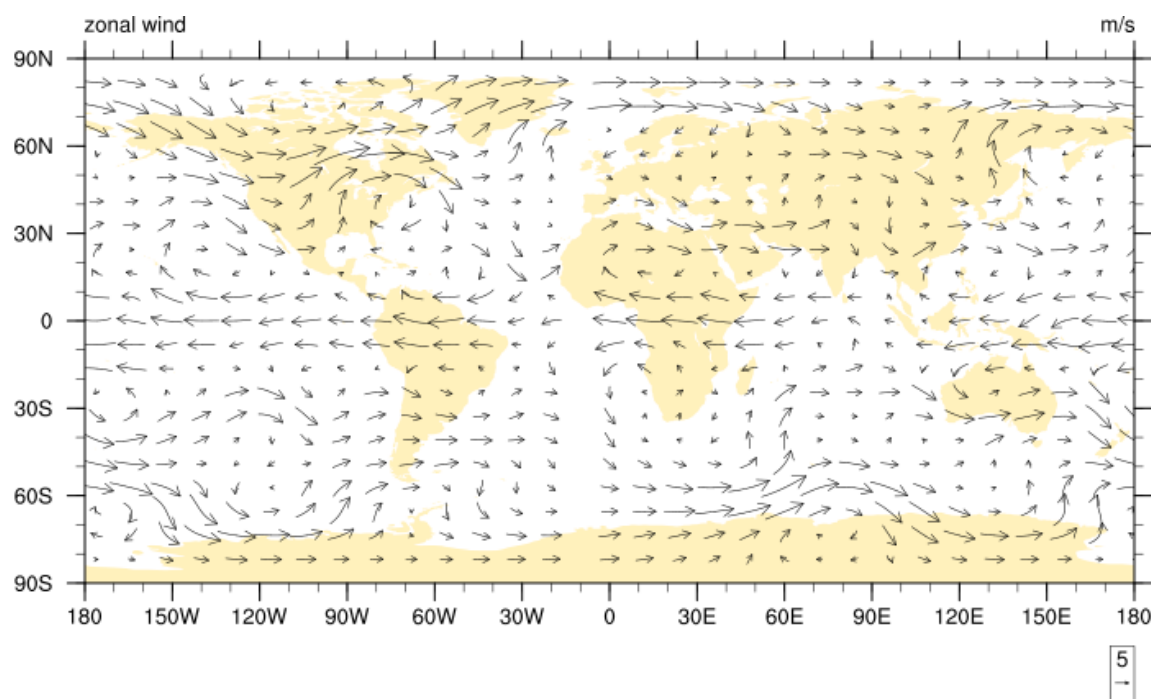
Example call:

```
ncl icon_plot.ncl 'varName="T"' 'iFile="iFILENAME"' \
  'secLC=(/0,80/)' 'secRC=(/0,-80/)'
```

`secPoints` is an option to set the accuracy of the plot. The representing of the location of the section is suppressed by setting `showSecMap=False`. Its default value is `True`.

### Display the ICON grid

Set the parameter `showGrid` to `True` and for scalar variables, the ICON grid is represented instead of the contour plot. For large grids, this can take a long time.



Prgr icon\_plot.ncl: Wed Nov 30 14:39:10 CET 2011,ram

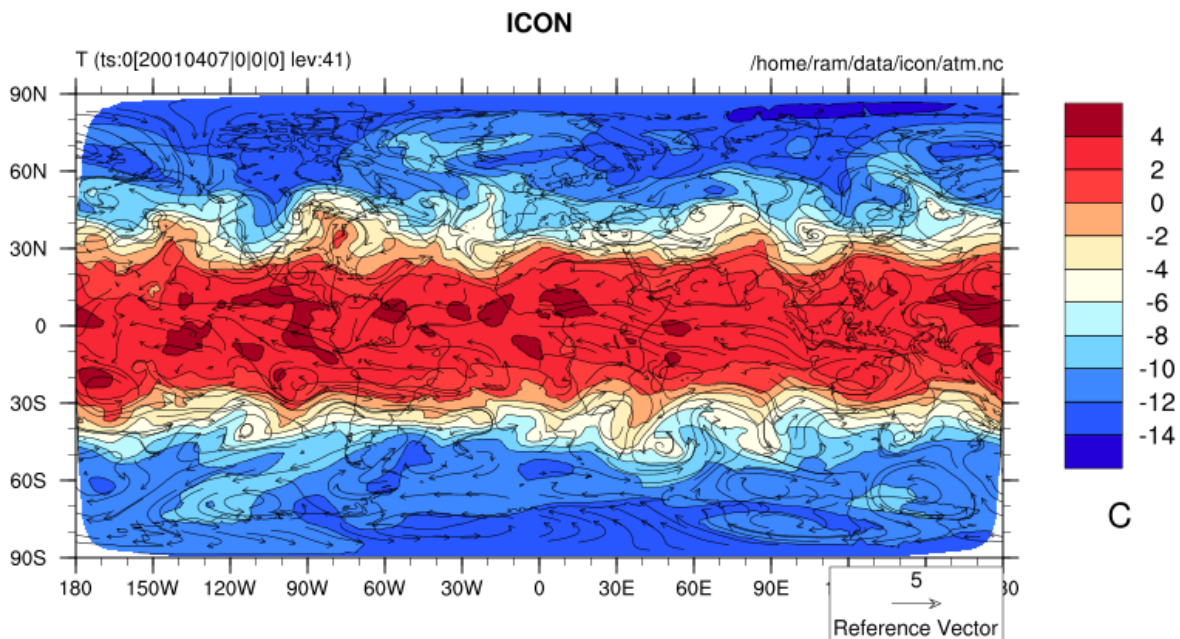
Figure 3.2: Example of vector plot

### 3.1.5 Regional plots

Use the variables `mapLLC` (map-Lower-Left-Corner) and `mapURC` (map-Upper-Right-Corner) to select special regions of the earth. Here is a list of useful examples:

Table 3.1: Examples of useful regional plots

Trop. Atlantic	'mapLLC=(/-60, -25/)'	'mapURC=(/ 25,25/)'
North Polar	'mapLLC=(/-200, 20/)'	'mapURC=(/160,90/)'
North Atlantic	'mapLLC=(/-100, -15/)'	'mapURC=(/ 35,65/)'
Labrador/Panama	'mapLLC=(/-200, -5/)'	'mapURC=(/ 35,85/)'
North Atlantic/Eurasia	'mapLLC=(/ -80, -5/)'	'mapURC=(/ 75,85/)'
Asia	'mapLLC=(/ 20, -15/)'	'mapURC=(/160,85/)'



Prgr icon\_plot.ncl: Wed Nov 30 14:51:24 CET 2011,ram

Figure 3.3: Example of overlay plot

### 3.1.6 Masking

Masking can be done in two different ways:

1. Manually mask the data with CDO before running the plot scripts, i.e. use the ifthen operator or perform a division with the mask variable:

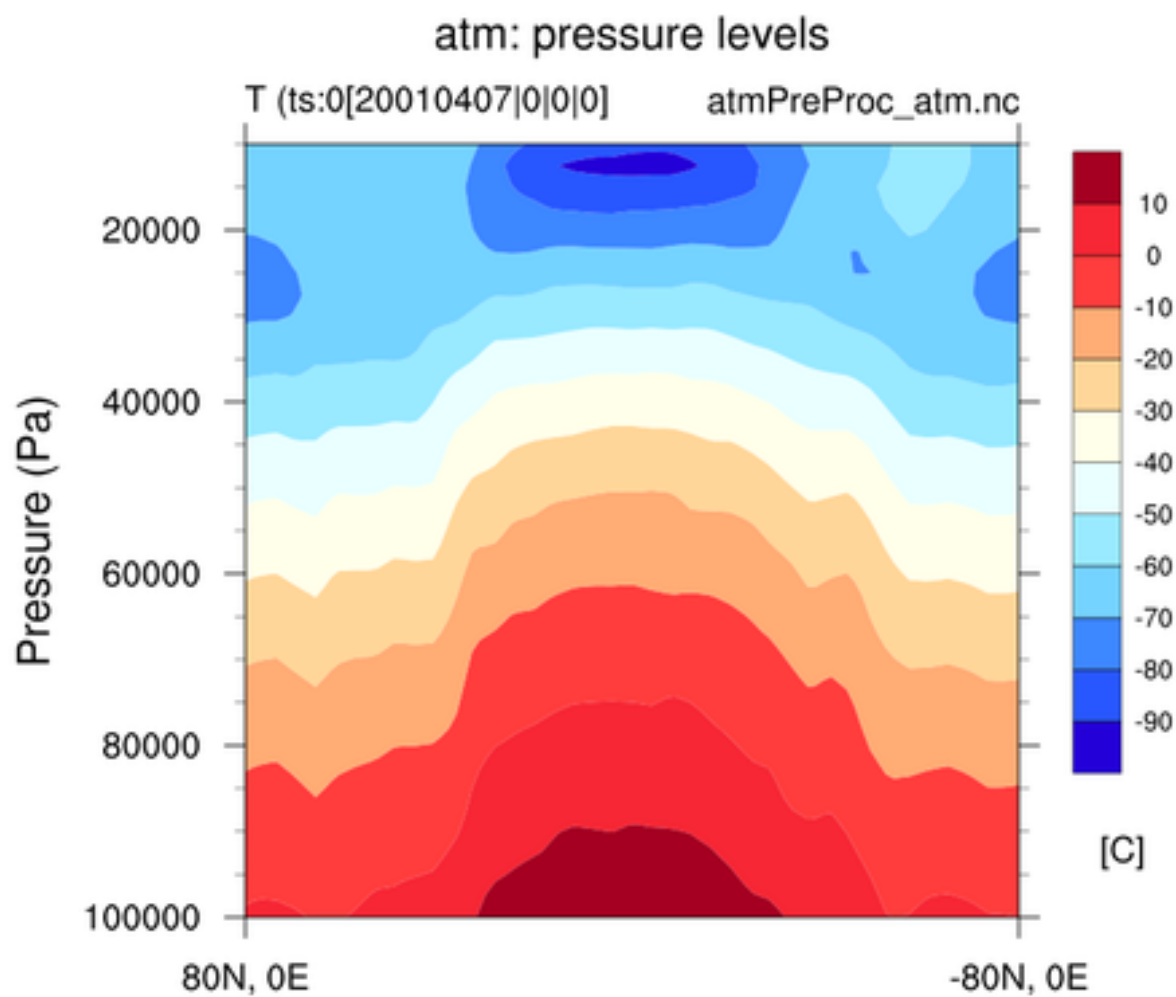
```
cdo div iconInput.nc -selname,mask_variable iconInput.nc maskedOutput.nc
```

2. Let the plot script perform the masking using the NCL's mask function. For this purpose, the commandline variables `maskName` and `maskFile` have to be used. If the mask variable is part of the regular input file, `maskFile` can be left out.

Both methods have their pros and cons. Whereas the second methods works fine for all types of horizontal representation, the first produces better results for vertical cross sections.

### 3.1.7 Data on other grids

Although `icon_plot.ncl` is implemented for ICON, it can be used for data on regular grids, too. In this case, internal interpolation is not performed.



Prgr icon\_plot.ncl: Wed Nov 30 15:05:50 CET 2011,ram

Figure 3.4: Example of vertical sections plot

### 3.1.8 All options

icon\_plot.ncl has built-in documentation of all options. Use

```
ncl icon_plot.ncl help=True
```

## 3.2 ncview/GrADS

Ncview ([http://meteora.ucsd.edu/~pierce/ncview\\_home\\_page.html](http://meteora.ucsd.edu/~pierce/ncview_home_page.html)) and GrADS (<http://www.iges.org/grads/>) can be used after converting icon data sets to a regular grid. This can easily be done with cdo:

```
cdo -P 8 -r remapnn,r180x90 icon.nc regular_icon.nc
```

This uses nearest neighbor interpolation and hereby keeps the model values. When using a higher regular resolution the triangular icon grid keeps visible.

### 3.3 Other Possibilities

- GMT is useful, when the grid should be visualized.
- ParaView is an alternative to display data on an unstructured grid. As a caveat, the model output has first to be converted into the vtk format.

**Discussion**

Document last edited by *I. Kraut* on *29.11.2013*.

Note: -

## 4 ICON Namelists Overview

### 4.1 Namelist Annotation

Every ICON run generates annotated lists of namelist parameters during the setup. These lists are written to text files `nml.atmo.log`, `nml.cpl.log`, `nml.ocean.log` and have the following form:

```
NAMELIST IO_NML
  OUT_EXPNAME      'case4'                [...] (truncated)
                  >> DEFAULT: 'IIIIIEETTTT' [...] (truncated)
  OUT_FILETYPE     2
  LKEEP_IN_SYNC    F
  DT_DATA          43200.000000000000
                  >> DEFAULT: 21600.000000000000
  DT_DIAG          1728000.0000000000
                  >> DEFAULT: 86400.000000000000
```

and so on.

The **DEFAULT** annotation denotes all those parameters that have been modified by the user; in this case the default value of the namelist parameter is stated together with the modified value. All other namelist parameters are listed only with their default values.



4.2 ICON Namelists

4.2.1 Scripts, Namelist files and Programs

Run scripts starting the programs for the grid generation and the models are stored in run/. These scripts write namelist files containing the specified Fortran namelists. Programs are stored in < icon home>/build/<architecture>/bin/.

Table 4.1: Namelist files

Namelist file	Purpose	Made by script	Used by program
NAMELIST_GRAPH	Generate graphs	create_global_grids.run	grid_command
NAMELIST_GRID	Generate grids	create_global_grids.run	grid_command
NAMELIST_GRIDREF	Gen. nested domains	create_global_grids.run	grid_command
NAMELIST_ICON	Run ICON models	expiname <sub>i</sub> .run	control_model

4.2.2 Namelist parameters

The following subsections tabulate all available Fortran namelist parameters by name, type, default value, unit, description, and scope:

- *Type* refers to the type of the Fortran variable, in which the value is stored: I=INTEGER, L=LOGICAL, R=REAL, C=character string
- *Default* is the preset value, if defined, that is assigned to this parameter within the programs.
- *Unit* shows the unit of the control parameter, where applicable.
- *Description* explains in a few words the purpose of the parameter.
- *Scope* explains under which conditions the namelist parameter has any effect, if its scope is restricted to specific settings of other namelist parameters.

Information on the file, where the namelist is defined and used, is given at the end of each table.

4.3 Namelist parameters for grid generation

4.3.1 Namelist parameters defining the atmosphere grid

graph\_ini (NAMELIST\_GRAPH)

Parameter	Type	Default	Unit	Description	Scope
nroot	I	2		root subdivision of initial edges	
grid_levels	I	4		number of edge bisections following the root subdivision	
lplane	L	.FALSE.		switch for generating a double periodic planar grid. The root level consists of 8 triangles.	

Defined and used in: src/grid\_generator/mo\_io\_graph.f90

**grid\_ini (NAMELIST\_GRID)**

Parameter	Type	Default	Unit	Description	Scope
<b>nroot</b>	I	2		root subdivision of initial edges	
<b>grid_levels</b>	I	4		number of edge bisections following the root subdivision	
<b>lplane</b>	L	.FALSE.		switch for generating planar grid. The root level consists of 8 triangles.	
<b>lread_graph</b>	L	.FALSE.		switch for reading graph information from precomputed file; .TRUE. implies that the graph generator needs to be executed in advance	

Defined and used in: `src/grid_generator/mo_grid_levels.f90`

**grid\_options (NAMELIST\_GRID)**

Parameter	Type	Default	Unit	Description	Scope
<b>x_rot_angle</b>	R	0.0	deg	Rotation of the icosahedron about the x-axis (connecting the origin and [0°E, 0°N])	
<b>y_rot_angle</b>	R	0.0	deg	Rotation of the icosahedron about the y-axis (connecting the origin and [90°E, 0°N]), done after the rotation about the x-axis.	
<b>z_rot_angle</b>	R	0.0	deg	rotation of the icosahedron about the z-axis (connecting the origin and [0°E, 90°N]), done after the rotation about the y-axis.	
<b>itype_optimize</b>	I	4		Grid optimization type 0: no optimization 1: Heikes Randall 2: equal area 3: c-grid small circle 4: spring dynamics	

Parameter	Type	Default	Unit	Description	Scope
<b>lc_grid</b>	L	.FALSE.		C-grid constraint on last level	
<b>maxlev_optim</b>	I	100		Maximum grid level where the optimization is applied	i_type_optimize = 1 or 4
<b>beta_spring</b>	R	0.90		tuning factor for target grid length	i_type_optimize = 4

Defined and used in: `src/grid_generator/mo_grid_levels.f90`

#### plane\_options (NAMELIST\_GRID)

Parameter	Type	Default	Unit	Description	Scope
<b>tria_arc_km</b>	R	10.0	km	length of triangle edge on plane	lplane=.TRUE.

The number of grid points is generated by root level section and further bisections. The double periodic root level consists of 8 triangles. The spatial coordinates are  $-1 < x <= 1$ , and  $-\sqrt{3}/2 < y <= \sqrt{3}/2$ . Currently the planar option can only be used as an *f*-plane.

Defined and used in: `src/grid_generator/mo_grid_levels.f90`

#### gridref\_ini (NAMELIST\_GRIDREF)

Parameter	Type	Default	Unit	Description	Scope
<b>grid_root</b>	I	2		root subdivision of initial edges	
<b>start_lev</b>	I	4		number of edge bisections following the root subdivision	
<b>n_dom</b>	I	2		number of logical model domains, including the global one	
<b>n_phys_dom</b>	I	n_dom		number of physical model domains, may be larger than n_dom (in this case, domain merging is applied)	

Parameter	Type	Default	Unit	Description	Scope
<b>parent_id</b>	I(n_phys_dom-1)	i		ID of parent domain (first entry refers to first nested domain; needs to be specified only in case of more than one nested domain per grid level)	
<b>logical_id</b>	I(n_phys_dom-1)	i+1		logical grid ID of domain (first entry refers to first nested domain; needs to be specified only in case of domain merging, i.e. n_dom < n_phys_dom)	
<b>l_plot</b>	L	.FALSE.		produces GMT plots showing the locations of the nested domains	
<b>l_circ</b>	L	.FALSE.		Create circular (.T.) or rectangular (.F.) refined domains	
<b>l_rotate</b>	L	.FALSE.		Rotates center point into the equator in case of l_circ = .FALSE.	lcirc=.FALSE.
<b>write_hierarchy</b>	I	1		0: Output only computational grids 1: Output in addition parent grid of global model domain (required for computing physics on a reduced grid) 2: Output all grids back to level 0 (required for hierarchical search algorithms)	
<b>lsep_gridref_info</b>	L	.FALSE.		.TRUE.: write fields describing parent-child connectivities into separate grid files	
<b>uuid_sourcefile</b>	C(n_dom)	'EMPTY'		If specified, provides the names of existing grid files from which the uuid shall be copied. If a radiation grid is present, the first entry refers to this grid.	
<b>bdy_indexing_depth</b>	I	12		Number of cell rows along the lateral boundary of a model domain for which the refin_ctrl fields contain the distance from the lateral boundary; needs to be enlarged when lateral boundary nudging is required for one-way nesting	

Parameter	Type	Default	Unit	Description	Scope
<b>radius</b>	R(n_dom-1)	30.	deg	radius of nested domain (first entry refers to first nested domain; needs to be specified for each nested domain separately)	lcirc=.TRUE.
<b>hwidth_lon</b>	R(n_dom-1)	20.	deg	zonal half-width of refined domain (first entry refers to first nested domain; needs to be specified for each nested domain separately)	lcirc=.FALSE.
<b>hwidth_lat</b>	R(n_dom-1)	20.	deg	meridional half-width of refined domain (first entry refers to first nested domain; needs to be specified for each nested domain separately)	lcirc=.FALSE.
<b>center_lon</b>	R(n_dom-1)	30.	deg	center longitude of refined domain (first entry refers to first nested domain; needs to be specified for each nested domain separately)	
<b>center_lat</b>	R(n_dom-1)	90.	deg	center latitude of refined domain (first entry refers to first nested domain; needs to be specified for each nested domain separately)	

Defined and used in: `src/grid_generator/mo_gridrefinement.f90`

#### **gridref\_metadata (NAMELIST\_GRIDREF)**

Parameter	Type	Default	Unit	Description	Scope
<b>number_of_grid_used</b>	I(n_dom+1)	0		sets the number of grid used in the netcdf header; the number of entries must be n_dom+1 since the first number refers to the radiation grid	
<b>centre</b>	I	0		centre running the grid generator 78: EDZW (DWD) 252: MPIM	
<b>subcentre</b>	I	0		subcentre to be assigned by centre, usually 0	

Parameter	Type	Default	Unit	Description	Scope
outname_style	I	1		Output name style 1: Standard: <i>iconRXX_DOMXX.nc</i> 2: DWD: <i>icon_grid_XXXX_RXXBXX_X.nc</i>	

Defined and used in: src/grid\_generator/mo\_gridrefinement.f90

4.3.2 coupling\_mode\_nml

Parameter	Type	Default	Unit	Description	Scope
coupled_mode	L	.FALSE.		.TRUE.: if yac coupling routines have to be called	

Defined and used in: src/namelist/mo\_coupling\_nml.f90

4.4 Namelist parameters defining the atmospheric model

Namelist parameters for the ICON models are organized in several thematic Fortran namelists controlling the experiment, and the properties of dynamics, transport, physics etc.

4.4.1 diffusion\_nml

Parameter	Type	Default	Unit	Description	Scope
lhdiff_temp	L	.TRUE.		Diffusion on the temperature field	
lhdiff_vn	L	.TRUE.		Diffusion on the horizontal wind field	
lhdiff_w	L	.TRUE.		Diffusion on the vertical wind field	

Parameter	Type	Default	Unit	Description	Scope
hdiff_order	I	4 (hydro) 5 (NH)		Order of $\nabla$ operator for diffusion: -1: no diffusion 2: $\nabla^2$ diffusion 3: Smagorinsky $\nabla^2$ diffusion 4: $\nabla^4$ diffusion 5: Smagorinsky $\nabla^2$ diffusion combined with $\nabla^4$ background diffusion as specified via hdiff_efdt_ratio	Options 2, 24 and 42 are allowed only in the hydrostatic atm model (iequations = 1 or 2 in dynamics_nml).
lsmag_3d	L	.FALSE.		24 or 42: $\nabla^2$ diffusion from model top to a certain level (cf. k2_pres_max and k2_klev_max below); $\nabla^4$ for the lower levels. .TRUE.: Use 3D Smagorinsky formulation for computing the horizontal diffusion coefficient (recommended at mesh sizes finer than 1 km if the LES turbulence scheme is not used)	hdiff_order=3 or 5; itype_vn_diffu=1
itype_vn_diffu	I	1		Reconstruction method used for Smagorinsky diffusion: 1: u/v reconstruction at vertices only 2: u/v reconstruction at cells and vertices	iequations=3, hdiff_order=3 or 5
itype_t_diffu	I	2		Discretization of temperature diffusion: 1: $K_h \nabla^2 T$ 2: $\nabla \cdot (K_h \nabla T)$	iequations=3, hdiff_order=3 or 5
k2_pres_max	R	-99.	Pa	Pressure level above which $\nabla^2$ diffusion is applied.	hdiff_order = 24 or 42, and dynam- ics_nml:iequations = 1 or 2.
k2_klev_max	I	0		Index of the vertical level till which (from the model top) $\nabla^2$ diffusion is applied. If a positive value is specified for k2_pres_max, k2_klev_max is reset accordingly during the initialization of a model run.	hdiff_order = 24 or 42, and dynam- ics_nml:iequations = 1 or 2.



Parameter	Type	Default	Unit	Description	Scope
hdiff_efdt_ratio	R	1.0 (hydro) 36.0 (NH)		ratio of e-folding time to time step (or 2* time step when using a 3 time level time stepping scheme) (for triangular NH model, values above 30 are recommended when using hdiff_order=5)	iequations=3
hdiff_w_efdt_ratio	R	15.0		ratio of e-folding time to time step for diffusion on vertical wind speed	iequations=3 .AND. hdiff_order=4
hdiff_min_efdt_ratio	R	1.0		minimum value of hdiff_efdt_ratio near model top	
hdiff_tv_ratio	R	1.0		Ratio of diffusion coefficients for temperature and normal wind: $T : v_n$	n_dom>1
hdiff_multfac	R	1.0		Multiplication factor of normalized diffusion coefficient for nested domains	
<b>hdiff_smag_fac</b>	R	0.15 (hydro) 0.015 (NH)		Scaling factor for Smagorinsky diffusion	iequations=3

Defined and used in: `src/namelists/mo_diffusion_nml.f90`

#### 4.4.2 dynamics\_nml

This namelist is relevant if `run_nml:ldynamics=.TRUE.`

Parameter	Type	Default	Unit	Description	Scope
<b>iequations</b>	I	3		Equations and prognostic variables. Use positive indices for the atmosphere and negative indices for the ocean. 0: shallow water model 1: hydrostatic atmosphere, T 2: hydrostatic atm., $\theta$ -dp 3: non-hydrostatic atmosphere -1: hydrostatic ocean	

Parameter	Type	Default	Unit	Description	Scope
idiv_method	I	1		Method for divergence computation: 1: Standard Gaussian integral. Hydrostatic atm. model: for unaveraged normal components Non-hydrostatic atm. model: for averaged normal components 2: bilinear averaging of divergence Weight of central cell for divergence averaging Coriolis force Reference height of shallow water model used for linearization in the semi-implicit time stepping scheme	idiv_method = 2
divavg_cntrwgt	R	0.5			
lcoriolis	L	.TRUE.			
sw_ref_height	R	0.9* 2.94e4/g	m		

Defined and used in: `src/namelist/mo_dynamics_nml.f90`

#### 4.4.3 echam\_conv\_nml

Parameter	Type	Default	Unit	Description	Scope
<b>iconv</b>	I	1		Choice of cumulus convection scheme. 1: Nordeng scheme 2: Tiedtke scheme 3: hybrid scheme	iforcing = 2 .AND. lconv = .TRUE.
ncvmicro	I	0		Choice of convective microphysics scheme.	iforcing = 2 .AND. lconv = .TRUE.
lmfpen	L	.TRUE.		Switch on penetrative convection.	iforcing = 2 .AND. lconv = .TRUE.
lmfmid	L	.TRUE.		Switch on midlevel convection.	iforcing = 2 .AND. lconv = .TRUE.
lmfdd	L	.TRUE.		Switch on cumulus downdraft.	iforcing = 2 .AND. lconv = .TRUE.
lmfdudv	L	.TRUE.		Switch on cumulus friction.	iforcing = 2 .AND. lconv = .TRUE.

Parameter	Type	Default	Unit	Description	Scope
cmfau	R	10800.		Characteristic convective adjustment time scale.	iforcing = 2 .AND. lconv = .TRUE.
cmfctop	R	0.3		Fractional convective mass flux (valid range [0,1]) across the top of cloud	iforcing = 2 .AND. lconv = .TRUE.
cprcon	R	1.0e-4		Coefficient for determining conversion from cloud water to rain.	iforcing = 2 .AND. lconv = .TRUE.
cminbuoy	R	0.025		Minimum excess buoyancy.	iforcing = 2 .AND. lconv = .TRUE.
entrpen	R	1.0e-4		Entrainment rate for penetrative convection.	iforcing = 2 .AND. lconv = .TRUE.
dlev	R	3.e4	Pa	Critical thickness necessary for the onset of convective precipitation.	iforcing = 2 .AND. lconv = .TRUE.

Defined and used in: `src/namelist/mo_echam_conv_nml.f90`

#### 4.4.4 echam\_phy\_nml

Parameter	Type	Default	Unit	Description	Scope
<b>lrad</b>	L	.TRUE.		.TRUE. for radiation.	run_nml/forcing = 2
<b>dt_rad</b>	R	3600.	s	time interval for radiative transfer computation	run_nml/forcing = 2
<b>lvdiff</b>	L	.TRUE.		.TRUE. for vertical turbulent diffusion	run_nml/forcing = 2
<b>lconv</b>	L	.TRUE.		.TRUE. for cumulus convection	run_nml/forcing = 2
<b>lcond</b>	L	.TRUE.		.TRUE. for large scale condensation	run_nml/forcing = 2
<b>lgw_hines</b>	L	.TRUE.		.TRUE. for non-orographic gravity wave drag (Hines)	run_nml/forcing = 2
<b>lsodrag</b>	L	.TRUE.		.TRUE. for subgrid scale orographic effects (Lott and Miller)	run_nml/forcing = 2
<b>lice</b>	L	.FALSE.		.TRUE. for sea-ice temperature calculation	run_nml/forcing = 2
<b>lmlo</b>	L	.FALSE.		.TRUE. for mixed layer ocean	run_nml/forcing = 2
<b>ljsbach</b>	L	.FALSE.		.TRUE. for calculating land surface properties (JSBACH)	run_nml/forcing = 2
<b>lamip</b>	L	.FALSE.		.TRUE. for AMIP boundary conditions	run_nml/forcing = 2

Defined and used in: `src/namelist/mo_echam_phy_nml.f90`

#### 4.4.5 ensemble\_pert\_nml

Parameter	Type	Default	Unit	Description	Scope
<b>use_ensemble_pert</b>	L	.FALSE.		Main switch to activate physics parameter perturbations for ensemble forecasts / ensemble data assimilation; the perturbations are applied via random numbers depending on the perturbationNumber (ensemble member ID) specified in gribout_nml	run_nml:forcing = inwp
<b>range_gkwake</b>	R	0.333		Variability range for low level wake drag constant	

Parameter	Type	Default	Unit	Description	Scope
range_gkdrag	R	0.04		Variability range for orographic gravity wave drag constant	
range_gfluxlaun	R	0.75e-3		Variability range for non-orographic gravity wave launch momentum flux	
range_zvz0i	R	0.2	m/s	Variability range for terminal fall velocity of ice	inwp_gscp = 1 or 2
range_entrorg	R	0.2e-3	1/m	Variability range for entrainment parameter in convection scheme	inwp_convection = 1
range_capdcfac.et	R	0.75		Maximum fraction of CAPE diurnal cycle correction applied in the extratropics	icapdcycl = 3
range_rhebc	R	0.05		Variability range for RH threshold for the onset of evaporation below cloud base	inwp_convection = 1
range_texc	R	0.05	K	Variability range for temperature excess value in test parcel ascent	inwp_convection = 1
range_box_liq	R	0.01		Variability range for box width scale of liquid clouds in cloud cover scheme	inwp_cldcover = 1
range_tkhmin	R	0.2		Variability range for minimum vertical diffusion for heat/moisture	inwp_turb = 1
range_tkmmin	R	0.2		Variability range for minimum vertical diffusion for momentum	inwp_turb = 1
range_tkred_sfc	R	4.0		Range for multiplicative change of reduction of minimum diffusion coefficients near the surface	inwp_turb = 1
range_rlam_heat	R	3.0		Variability range (multiplicative!) of laminar transport resistance parameter	inwp_turb = 1
range_chnock	R	1.5		Variability range (multiplicative!) of upper and lower bound of wind-speed dependent Charnock parameter	inwp_turb = 1
range_minsnowfrac	R	0.05		Variability range for minimum value to which snow cover fraction is artificially reduced in case of melting snow	idiag_snowfrac = 20/30/40
range_z0_lcc	R	0.25		Variability range (relative change) of roughness length attributed to each landuse class	

Parameter	Type	Default	Unit	Description	Scope
range_rootdp	R	0.2		Variability range (relative change) of root depth attributed to each landuse class	
range_rsmin	R	0.2		Variability range (relative change) of minimum stomata resistance attributed to each landuse class	
range_laimax	R	0.15		Variability range (relative change) of leaf area index (maximum of annual cycle) attributed to each landuse class	

Defined and used in: `src/namelist/mo_ensemble_pert_nml.f90`

#### 4.4.6 gribout\_nml

Parameter	Type	Default	Unit	Description	Scope
preset	C	"determ.."		Setting this different to "none" enables a couple of defaults for the other <code>gribout_nml</code> namelist parameters. If, additionally, the user tries to set any of these other parameters to a conflicting value, an error message is thrown. Possible values are "none", "deterministic", "ensemble".	filetype=2
backgroundProcess	I	0		Background process - GRIB2 code table <code>backgroundProcess.table</code>	filetype=2
generatingCenter	I	-1		Output generating center. If this key is not set, center information is taken from the grid file DWD: 78 MPIMET: 98 ECMWF: 98	filetype=2

Parameter	Type	Default	Unit	Description	Scope
generatingSubcenter	I	-1		Output generating Subcenter. If this key is not set, subcenter information is taken from the grid file DWD: 255 MPIMET: 232 ECMWF: 0	filetype=2
generatingProcessIdentifier	I(n_dom)	1		generating Process Identifier - GRIB2 code table	filetype=2
numberOfForecastsInEnsemble	I	-1		generatingProcessIdentifier.table Local definition for ensemble products, (only set if value changed from default)	filetype=2
perturbationNumber	I	-1		Local definition for ensemble products, (only set if value changed from default)	filetype=2
productionStatusOfProcessedData	I	1		Production status of data - GRIB2 code table 1.3	filetype=2
significanceOfReferenceTime	I	1		Significance of reference time - GRIB2 code table 1.2	filetype=2
typeOfEnsembleForecast	I	-1		Local definition for ensemble products (only set if value changed from default)	filetype=2
typeOfGeneratingProcess	I	-1		Type of generating process - GRIB2 code table 4.3	filetype=2
typeOfProcessedData	I	-1		Type of data - GRIB2 code table 1.4	filetype=2
localDefinitionNumber	I	-1		local Definition Number - GRIB2 code table	filetype=2
localNumberOfExperiment	I	1		grib2LocalSectionNumber.78.table local Number of Experiment	filetype=2
localTypeOfEnsembleForecast	I	-1		Local definition for ensemble products (only set if value changed from default)	filetype=2

Parameter	Type	Default	Unit	Description	Scope
lspecialdate_invar	L	.FALSE.		Special reference date for invariant and climatological fields .TRUE.: set special reference date 0001-01-01, 00:00 .FALSE.: no special reference date	filetype = 2
ldate_grib_act	L	.TRUE.		GRIB creation date .TRUE.: add creation date .FALSE.: add dummy date	filetype=2
lgribout_24bit	L	.FALSE.		If TRUE, write thermodynamic fields $\rho$ , $\theta_v$ , $T$ , $p$ with 24bit precision instead of 16bit	filetype=2

Defined and used in: `src/namelist/mo_gribout_nml.f90`

#### 4.4.7 grid\_nml

Parameter	Type	Default	Unit	Description	Scope
cell_type	I	3		Cell type: not used	
lplane	L	.FALSE.		planar option	
is_plane_torus	L	.FALSE.		f-plane approximation on triangular grid	
corio_lat	R	0.0	deg	Center of the f-plane is located at this geographical latitude	lplane=.TRUE. and is_plane_torus=.TRUE.
grid_angular_velocity	R	Earth's	rad/s	The angular velocity in rad per sec.	
<b>l_limited_area</b>	L	.FALSE.		Defined as the inverse of the reduced-size earth reduction factor $X$ . Choose	
grid_rescale_factor	R	1.0		<b>grid_rescale_factor</b> < 1 for a reduced-size earth. The geometry and the timestep will be multiplied by this factor. The angular velocity will be divided by this factor.	



Parameter	Type	Default	Unit	Description	Scope
<b>lfeedback</b>	L(n_dom)	.TRUE.		Specifies if feedback to parent grid is performed. Setting lfeedback(1)=.false. turns off feedback for all nested domains; to turn off feedback for selected nested domains, set lfeedback(1)=.true. and set ".false." for the desired model domains	n_dom>1
ifedback_type	I	2		1: incremental feedback 2: relaxation-based feedback Note: vertical nesting requires option 2 to run numerically stable over longer time periods	n_dom>1
start_time	R(n_dom)	0.	s	Time when a nested domain starts to be active (namelist entry is ignored for the global domain)	n_dom>1
end_time	R(n_dom)	1.E30	s	Time when a nested domain terminates (namelist entry is ignored for the global domain)	n_dom>1
patch_weight	R(n_dom)	0.		If patch_weight is set to a value > 0 for any of the first level child patches, processor splitting will be performed, i.e. every of the first level child patches gets a subset of the total number of processors corresponding to its patch_weight. A value of 0. corresponds to exactly 1 processor for this patch, regardless of the total number of processors. For the root patch and higher level childs, patch_weight is not used. However, patch_weight must be set to 0 for these patches to avoid confusion.	n_dom>1
iredgrid.phys	L	.FALSE.		If set to .true. radiation is calculated on a reduced grid (= one grid level higher)	
dynamics_grid_filename	C			Array of the grid filenames to be used by the dycore. May contain the keyword <path> which will be substituted by model_base_dir.	

Parameter	Type	Default	Unit	Description	Scope
<b>dynamics_parent_grid_id</b>	I(n_dom)	$i - 1$		Array of the indexes of the parent grid filenames, as described by the <code>dynamics_grid_filename</code> array. Indexes start at 1, an index of 0 indicates no parent.	
<b>radiation_grid_filename</b>	C			Array of the grid filenames to be used for the radiation model. Filled only if the radiation grid is different from the dycore grid. May contain the keyword <code>&lt;path&gt;</code> which will be substituted by <code>model_base_dir</code> .	<code>lredgrid_phys=.TRUE.</code>
<b>dynamics_radiation_grid_link</b>	I(n_dom)	1 for $i=1$		Array of the indexes linking the dycore grids, as described by the <code>dynamics_grid_filename</code> array, and the <code>radiation_grid_filename</code> array. It provides the link index of the <code>radiation_grid_filename</code> , for each entry of the <code>dynamics_grid_filename</code> array. Indexes start at 1, an index of 0 indicates that the radiation grid is the same as the dycore grid. Only needs to be filled when the <code>radiation_grid_filename</code> is defined.	
<b>create_vgrid</b>	L	<code>.FALSE.</code>		<code>.TRUE.:</code> Write vertical grid files containing ( <code>vct_a</code> , <code>vct_b</code> , <code>z_ifc</code> , and <code>z_ifv</code> ).	
<b>vertical_grid_filename</b>	C(n_dom)			Array of filenames. These files contain the vertical grid definition ( <code>vct_a</code> , <code>vct_b</code> , <code>z_ifc</code> ). If empty, the vertical grid is created within ICON during the setup phase.	
<b>use_duplicated_connectivity</b>	L	<code>.TRUE.</code>		if <code>.TRUE.</code> , the zero connectivity is replaced by the last non-zero value	
<b>use_dummy_cell_closure</b>	L	<code>.FALSE.</code>		if <code>.TRUE.</code> then create a dummy cell and connect it to cells and edges with no neighbor	

Defined and used in: `src/namelist/mo_grid_nml.f90`

## 4.4.8 gridref\_nml

Parameter	Type	Default	Unit	Description	Scope
grf_intmethod_c	I	2		Interpolation method for grid refinement (cell-based dynamical variables): 1: parent-to-child copying 2: gradient-based interpolation	n_dom > 1
grf_intmethod_ct	I	2		Interpolation method for grid refinement (cell-based tracer variables): 1: parent-to-child copying 2: gradient-based interpolation	n_dom > 1
grf_intmethod_e	I	6		Interpolation method for grid refinement (edge-based variables): 1: inverse-distance weighting (IDW) 2: RBF interpolation 3: combination gradient-based / IDW 4: combination gradient-based / RBF 5/6: same as 3/4, respectively, but direct interpolation of mass fluxes along nest interface edges	n_dom > 1
grf_velfbk	I	1		Method of velocity feedback: 1: average of child edges 1 and 2	n_dom > 1
grf_scalfbk	I	2		2: 2nd-order method using RBF interpolation Feedback method for dynamical scalar variables ( $T, p_{sc}$ ): 1: area-weighted averaging 2: bilinear interpolation	n_dom > 1
grf_tracfbk	I	2		Feedback method for tracer variables: 1: area-weighted averaging 2: bilinear interpolation	n_dom > 1
grf_idw_exp_e12	R	1.2		exponent of generalized IDW function for child edges 1/2	n_dom > 1
grf_idw_exp_e34	R	1.7		exponent of generalized IDW function for child edges 3/4	n_dom > 1

Parameter	Type	Default	Unit	Description	Scope
rbf_vec_kern_grf_e	I	1		RBF kernel for grid refinement (edges): 1: Gaussian 2: $1/(1 + r^2)$ 3: inverse multiquadric	n_dom>1
rbf_scale_grf_e	R(n_dom)	0.5		RBF scale factor for grid refinement (lateral boundary interpolation to edges). Refers to the respective parent domain and thus does not need to be specified for the innermost nest. Lower values than the default of 0.5 are needed for child mesh sizes less than about 500 m. Denominator for lateral boundary diffusion of temperature Denominator for lateral boundary diffusion of velocity	n_dom>1
denom_diffu_t	R	135		.TRUE.: Apply mass conservation correction in feedback routine	n_dom>1
denom_diffu_v	R	200		.TRUE.: Apply density nudging near lateral nest boundary if grf.intmethod_e ≤ 4	n_dom>1
l_mass_consvcorr	L	.FALSE.		Relaxation time scale for feedback	n_dom>1 .AND. lfeedback = .TRUE.
l_density_nudging	L	.FALSE.			n_dom>1 .AND. lfeedback = .TRUE. .AND. iffeedback_type = 2
fbk_relax_timescale	R	10800			

Defined and used in: src/namelist/mo\_gridref\_nml.f90

#### 4.4.9 gw\_hines\_nml (Scope: lgw\_hines = .TRUE. in echam\_phy\_nml)

Parameter	Type	Default	Unit	Description	Scope
lheatcal	L	.FALSE.		.TRUE.: compute drag, heating rate and diffusion coefficient from the dissipation of gravity waves	

Parameter	Type	Default	Unit	Description	Scope
emiss_lev	I	10		.FALSE.: compute drag only Index of model level, counted from the surface, from which the gravity wave spectra are emitted	
rmscon	R	1.0	m/s	Root mean square gravity wave wind at the emission level	
kstar	R	5.0e-5	1/m	Typical gravity wave horizontal wavenumber	
m_min	R	0.0	1/m	Minimum bound in vertical wavenumber	
lrmscon_lat	L	.FALSE.		.TRUE.: use latitude dependent rms wind - —latitude— >= lat_rmscon: use rmscon - —latitude— <= lat_rmscon_eq: use rmscon_eq - lat_rmscon_eq < —latitude— < lat_rmscon: use linear interpolation between rmscon_eq and rmscon	
lat_rmscon_eq	R	5.0	deg N	.FALSE.: use globally constant rms wind rmscon	lrmscon_lat = .TRUE.
lat_rmscon	R	10.0	deg N	rmscon_eq is used equatorward of this latitude rmscon is used poleward of this latitude	lrmscon_lat = .TRUE.
rmscon_eq	R	1.2	m/s	is used equatorward of latitude lat_rmscon_eq	lrmscon_lat = .TRUE.

Defined and used in: `src/namelist/mo_gw_hines_nml.f90`

#### 4.4.10 ha\_dyn\_nml

This namelist is relevant if `run_nml:ldynamics=.TRUE.` and `dynamics_nml:iequations=IHS_ATM_TEMP` or `IHS_ATM_THETA`.

Parameter	Type	Default	Unit	Description	Scope
<b>itime_scheme</b>	I	14		Time integration scheme: 11: pure advection (no dynamics) 12: 2 time level semi implicit (not yet implemented)	

Parameter	Type	Default	Unit	Description	Scope
ileapfrog_startup	I	1		13: 3 time level explicit 14: 3 time level with semi implicit correction 15: standard 4th-order Runge-Kutta method (4-stage) 16: SSPRK(5,4) scheme (5-stage) How to integrate the first time step when the leapfrog scheme is chosen. 1 = Euler forward; 2 = a series of sub-steps.	itime_scheme= 13 or 14
asselin_coeff	R	0.1		Asselin filter coefficient	itime_scheme= 13 or 14
si_2tls	R	0.6		weight of time step n+1. Valid range: [0,1]	itime_scheme=12
si_expLscheme	I	2		scheme for the explicit part used in the 2 time level semi-implicit time stepping scheme. 1 = Euler forward; 2 = Adams-Bashforth 2nd order	itime_scheme=12
si_cmin	R	30.0	m/s	semi implicit correction is done for eigenmodes with speeds larger than si_cmin	itime_scheme=14 and lsi_3d=.FALSE.
si_coeff	R	1.0		weight of the semi implicit correction	itime_scheme=14
si_offctr	R	0.7			itime_scheme=14
si_rtol	R	1.0e-3		relative tolerance for GMRES solver	itime_scheme=14
lsi_3d	L	.FALSE.		3D GMRES solver or decomposition into 2D problems	lshallow_water=.FALSE. and itime_scheme=14
ldry_dycore	L	.TRUE.		Assume dry atmosphere	iequations∈{1,2}
lref_temp	L	.FALSE.		Set a background temperature profile as base state when computing the pressure gradient force	iequations∈{1,2}

#### 4.4.11 initicon\_nml

Parameter	Type	Default	Unit	Description	Scope
<b>init_mode</b>	I	2		<p>1: MODE_DWDANA start from DWD analysis or FG</p> <p>2: MODE_IFSANA start from IFS analysis</p> <p>3: MODE_COMBINED IFS atm + ICON/GME soil</p> <p>4: MODE_COSMODE start from COSMO-DE forecast</p> <p>5: MODE_IAU start from DWD analysis with incremental analysis update. Extension of MODE_IAU_OLD including snow increments</p> <p>6: MODE_IAU_OLD start from DWD analysis with incremental analysis update. NOTE: Extension of mode MODE_DWDANA_INC including W_SO increments.</p> <p>7: MODE_ICONVREMAP start from DWD first guess with subsequent vertical remapping (work in progress; so far, changing the number of model levels does not yet work)</p>	init_mode=5,6  init_mode=5,6
dt_iau	R	10800	s	Time interval during which an incremental analysis update (IAU) is performed	
dt_shift	R	0	s	Time by which the actual model start time is shifted ahead of the nominal date. Must be NEGATIVE, usually $-0.5 \text{ dt\_iau}$ .	
start_time_avg_fg	R	0	s	Start time for calculating temporally averaged first guess output for data assimilation.	
end_time_avg_fg	R	0	s	End time for calculating temporally averaged first guess output for data assimilation. Setting end_time_avg_fg > start_time_avg_fg activates the averaging	

Parameter	Type	Default	Unit	Description	Scope
interval_avg_fg	R	0	s	Corresponding averaging interval. Note that end_time_avg_fg – start_time_avg_fg must not be smaller than the averaging interval	
rho_incr_filter_wgt	R	0		Vertical filtering weight on density increments	init_mode=5,6
type_iau_wgt	I	1		Weighting function for performing IAU 1: Top-Hat 2: SIN2	init_mode=5,6
nlevsoil_in	I	4		number of soil levels of input data	init_mode=2
zpbl1	R	500.0	m	bottom height (AGL) of layer used for gradient computation	
zpbl2	R	1000.0	m	top height (AGL) of layer used for gradient computation	
l_sst_in	L	.TRUE.		Logical switch. If true, the surface temperature of the water sea points is initialized with the SST provided in the ifs2icon file. If false, it is initialized with the skin temperature. If the SST is not provided in the ifs2icon file, l_sst_in is reset to false.	init_mode=2
lread_ana	L	.TRUE.		If .FALSE., ICON is started from first guess only. Analysis field is not required, and skipped if provided.	init_mode=1,3
lconsistency_checks	L	.TRUE.		If .FALSE., consistency checks for Analysis and First Guess fields are skipped. On default, checks are performed for <i>uwindOfHGrid</i> and <i>validity time</i> .	init_mode=1,3,4,5,6
l_coarse2fine_mode	L(n_dom)	.FALSE.		If true, apply corrections for coarse-to-fine mesh interpolation to wind and temperature	
lp2cintp_incr	L(n_dom)	.FALSE.		If true, interpolate atmospheric data assimilation increments from parent domain. Can be specified separately for each nested domain; setting the first (global) entry to true activates the interpolation for all nested domains.	init_mode=5,6



Parameter	Type	Default	Unit	Description	Scope
<code>lp2cintp_sfcana</code>	L(n.dom)	.FALSE.		If true, interpolate atmospheric surface analysis data from parent domain. Can be specified separately for each nested domain; setting the first (global) entry to true activates the interpolation for all nested domains.	init_mode=5,6
<code>ltile_init</code>	L	.FALSE.		True: initialize tiled surface fields from a first guess coming from a run without tiles. Along coastlines and lake shores, a neighbor search is executed to fill the variables on previously non-existing land or water points with reasonable values. Should be combined with <code>ltile_coldstart = .TRUE.</code>	init_mode=1,5,6
<code>ltile_coldstart</code>	L	.FALSE.		If true, tiled surface fields are initialized with tile-averaged fields from a previous run with tiles.	init_mode=1,5,6
<code>lvert_remap_fg</code>	L	.FALSE.		A neighbor search is applied to subgrid-scale ocean points for SST and sea-ice fraction. If true, vertical remapping is applied to the atmospheric first-guess fields, whereas the analysis increments remain unchanged. The number of model levels must be the same for input and output fields, and the <code>z_1fc</code> (alias HHL) field pertaining to the input fields must be appended to the first-guess file.	init_mode=5,6
<code>ifs2icon_filename</code>	C			Filename of IFS2ICON input file, default " <code>&lt;path&gt;ifs2icon.R&lt;nroot&gt;B&lt;jlev&gt;_DOM&lt;idom&gt;.nc</code> ". May contain the keywords <code>&lt;path&gt;</code> which will be substituted by <code>model_base_dir</code> , as well as <code>nroot</code> , <code>jlev</code> , and <code>idom</code> defining the current patch.	init_mode=2

Parameter	Type	Default	Unit	Description	Scope
<b>dwdfg_filename</b>	C			Filename of DWD first-guess input file, default " <code>&lt;path&gt;dwdFG_R&lt;nroot&gt;B&lt;jlev&gt;_DOM&lt;idom&gt;.nc</code> ". May contain the keywords <code>&lt;path&gt;</code> which will be substituted by <code>model_base_dir</code> , as well as <code>nroot</code> , <code>jlev</code> , and <code>idom</code> defining the current patch.	init_mode=1,3,5,6
<b>dwdana_filename</b>	C			Filename of DWD analysis input file, default " <code>&lt;path&gt;dwdana_R&lt;nroot&gt;B&lt;jlev&gt;_DOM&lt;idom&gt;.nc</code> ". May contain the keywords <code>&lt;path&gt;</code> which will be substituted by <code>model_base_dir</code> , as well as <code>nroot</code> , <code>jlev</code> , and <code>idom</code> defining the current patch.	init_mode=1,3,5,6
<b>filetype</b>	I	-1 (undef.)		One of CDF's FILETYPE_XXX constants. Possible values: 2 (=FILETYPE_GRB2), 4 (=FILETYPE_NC2). If this parameter has not been set, we try to determine the file type by its extension <code>"*.grb"</code> or <code>".nc"</code> .	
<b>ana_varlist</b>	C(:)			List of mandatory analysis fields for the <b>global domain</b> that must be present in the analysis file. If these fields are not found, the model aborts. For all other analysis fields, the FG-fields will serve as fallback position.	init_mode=1,5,6
<b>ana_varlist_n2</b>	C(:)			List of mandatory analysis fields for <b>domain 2</b> that must be present in the analysis file. If these fields are not found, the model aborts. For all other analysis fields, the FG-fields will serve as fallback position.	init_mode=5,6
<b>ana_varnames_map_file</b>	C			Dictionary file which maps internal variable names onto GRIB2 shortnames or NetCDF var names. This is a text file with two columns separated by whitespace, where left column: ICON variable name, right column: GRIB2 short name.	

Parameter	Type	Default	Unit	Description	Scope
<b>latbc_varnames_map_file</b>	C			Dictionary file which maps internal variable names onto GRIB2 shortnames or NetCDF variable names. This is a text file with two columns separated by whitespace, where left column: ICON variable name, right column: GRIB2 short name. This list contains variables that are to be read asynchronously for boundary data nudging in a HDCP2 simulation. All new boundary variables that in the future, would be read asynchronously. Need to be added to text file dict.latbc in run folder.	num_prefetch_proc=1

Defined and used in: `src/namelist/mo_init/icon_nml.f90`

#### 4.4.12 interpola\_nml

Parameter	Type	Default	Unit	Description	Scope
<code>l_intra_c2l</code>	L	.TRUE.		DEPRECATED	
<code>l_mono_c2l</code>	L	.TRUE.		Monotonicity can be enforced by demanding that the interpolated value is not higher or lower than the stencil point values.	
<code>llsq_high_consv</code>	L	.TRUE.		conservative (T) or non-conservative (F) least-squares reconstruction for high order transport	
<code>llsq_high_ord</code>	I	3		polynomial order for high order reconstruction 1: linear 2: quadratic 30: cubic (no 3 <sup>rd</sup> order cross deriv.) 3: cubic	ihadv_tracer=4
<code>llsq_lin_consv</code>	L	.FALSE.		conservative (T) or non-conservative (F) least-squares reconstruction for 2nd order (linear) transport	

Parameter	Type	Default	Unit	Description	Scope
nudge_efold_width	R	2.0		e-folding width (in units of cell rows) for lateral boundary nudging coefficient	
nudge_max_coeff	R	0.02		Maximum relaxation coefficient for lateral boundary nudging	
nudge_zone_width	I	8		Total width (in units of cell rows) for lateral boundary nudging zone. If $\neq 0$ the patch boundary_depth_index is used.	
rbf_dim_c2l	I	10		stencil size for direct lon-lat interpolation: 4 = nearest neighbor, 13 = vertex stencil, 10 = edge stencil.	
rbf_scale_mode_ll	I	2		Specifies, how the RBF shape parameter is determined for lon-lat interpolation. 1 : lookup table based on grid level 2 : determine automatically. So far, this routine only estimates the smallest value for the shape parameter for which the Cholesky is likely to succeed in floating point arithmetic. 3 : explicitly set shape parameter in each output namelist	
rbf_vec_kern_c	I	1		Kernel type for reconstruction at cell centres: 1: Gaussian 3: inverse multiquadric	
rbf_vec_kern_e	I	3		Kernel type for reconstruction at edges: 1: Gaussian 3: inverse multiquadric	
rbf_vec_kern_ll	I	1		Kernel type for reconstruction at lon-lat-points: 1: Gaussian 3: inverse multiquadric	
rbf_vec_kern_v	I	1		Kernel type for reconstruction at vertices: 1: Gaussian 3: inverse multiquadric	
rbf_vec_scale_c	R(n_dom)	resolution-dependent		Scale factor for RBF reconstruction at cell centres	

Parameter	Type	Default	Unit	Description	Scope
rbf_vec_scale_e	R(n_dom)	resolution+ dependent		Scale factor for RBF reconstruction at edges	
rbf_vec_scale_v	R(n_dom)	resolution+ dependent		Scale factor for RBF reconstruction at vertices	
support_baryctr_intp	L	.FALSE.		Flag. If .FALSE. barycentric interpolation is replaced by a fallback interpolation.	

Defined and used in: `src/namelist/mo_interpol_nml.f90`

#### 4.4.13 io\_nml

Parameter	Type	Default	Unit	Description	Scope
lkeep_in_sync	L	.FALSE.		Sync output stream with file on disk after each timestep	
dt_diag	R	86400.	s	diagnostic integral output interval	<code>run_nml:output = "totint"</code>
<b>dt_checkpoint</b>	R	2592000	s	Time interval for writing restart files. Note that if the value of <code>dt_checkpoint</code> resulting from model default or user's specification is longer than <code>time_nml:dt_restart</code> , it will be reset (by the model) to <code>dt_restart</code> so that at least one restart file is generated during the restart cycle.	<code>output /= "none"</code> ( <code>run_nml</code> )
inextra_2d	I	0		Number of extra 2D Fields for diagnostic/debugging output.	<code>dynamics_nml:iequations = 3</code> (to be done for 1, 2)
inextra_3d	I	0		Number of extra 3D Fields for diagnostic/debugging output.	<code>dynamics_nml:iequations = 3</code> (to be done for 1, 2)

Parameter	Type	Default	Unit	Description	Scope
lflux_avg	L	.TRUE.		if .FALSE. the output fluxes are accumulated from the beginning of the run if .TRUE. the output fluxes are average values from the beginning of the run, except of TOT_PREC that would be accumulated	iequations=3 iforcing=3
itype_pres_msl	I	1		Specifies method for computation of mean sea level pressure (and geopotential at pressure levels below the surface). 1: GME-type extrapolation, 2: stepwise analytical integration, 3: current IFS method, 4: IFS method with consistency correction	
itype_rh	I	1		Specifies method for computation of relative humidity 1: WMO-type: water only (e_s=e_s_water), 2: IFS-type: mixed phase (water and ice), 3: IFS-type with clipping (rh ≤ 100)	
output_nml_dict	C	, ,		File containing the mapping of variable names to the internal ICON names. May contain the keyword <path> which will be substituted by <code>model_base_dir</code> . The format of this file: One mapping per line, first the name as given in the <code>ml_varlist</code> , <code>hl_varlist</code> , <code>pl_varlist</code> or <code>il_varlist</code> of the <code>output_nml</code> namelists, then the internal ICON name, separated by an arbitrary number of blanks. The line may also start and end with an arbitrary number of blanks. Empty lines or lines starting with # are treated as comments. Names not covered by the mapping are used as they are.	output_nml_namelists

Parameter	Type	Default	Unit	Description	Scope
netcdf_dict	C	, ,		File containing the mapping from internal names to names written to NetCDF. May contain the keyword <path> which will be substituted by <code>model_base_dir</code> . The format of this file: One mapping per line, first the name written to NetCDF, then the internal name, separated by an arbitrary number of blanks ( <i>inverse to the definition of</i> <code>output_nml_dict</code> ). The line may also start and end with an arbitrary number of blanks. Empty lines or lines starting with # are treated as comments. Names not covered by the mapping are output as they are. Note that the specification of output variables, e.g. in <code>ml_varlist</code> , is independent from this renaming, see the namelist parameter <code>output_nml_dict</code> for this.	output_nml namelists, NetCDF output
lnetcdf_ft64_output	L	.FALSE.		If .TRUE. floating point variable output in NetCDF files is written in 64-bit instead of 32-bit accuracy. This is currently implemented for the atm. dynamical core and ECHAM physics.	
restart_file_type	I	4		Type of restart file. One of CDI's FILETYPE_XXX. So far, only 4 (=FILETYPE_NC2) is allowed	
lmask_boundary	L	F		Set to .TRUE., if interpolation zone should be masked in output.	

Defined and used in: `src/namelists/mo_io_nml.f90`

#### 4.4.14 les\_nml (parameters for LES turbulence scheme; valid for inwp\_turb=5)

Parameter	Type	Default	Unit	Description	Scope
sst	R	300	K	sea surface temperature for idealized LES simulations	isrfc_type=5,4
shflux	R	0.1	Km/s	Kinematic sensible heat flux at surface	isrfc_type = 2
lhflux	R	0	m/s	Kinematic latent heat flux at surface	isrfc_type = 2
isrfc_type	I	1		surface type 0 = No fluxes and zero shear stress 1 = TERRA land physics 2 = fixed surface fluxes 3 = fixed buoyancy fluxes 4 = RICO test case 5 = fixed SST	
ufc	R	-999	m/s	friction velocity for idealized LES simulations; if j 0 then it is automatically diagnosed	
psfc	R	-999	Pa	surface pressure for idealized LES simulations; if j 0 then it uses the surface pressure from dynamics	
min_sfc_wind	R	1.0	m/s	Minimum surface wind for surface layer useful in the limit of free convection	
is_dry_cbl	L	.FALSE.		switch for dry convective boundary layer simulations	
smag_constant	R	0.23		Smagorinsky constant	
km_min	R	0.0		Minimum turbulent viscosity	
max_turb_scale	R	300.0		Asymptotic maximum turbulence length scale (useful for coarse grid LES and when grid is vertically stretched)	
turb_prandtl	R	0.333333		turbulent Prandtl number	
bflux	R	0.0007	m <sup>2</sup> /s <sup>3</sup>	buoyancy flux for idealized LES simulations (Stevens 2007)	isrfc_type=3
tran_coeff	R	0.02	m/s	transfer coefficient near surface for idealized LES simulation (Stevens 2007)	isrfc_type=3



Parameter	Type	Default	Unit	Description	Scope
vert_scheme_type	I	2		type of time integration scheme in vertical diffusion 1 = explicit 2 = fully implicit	
sampl_freq_sec	R	60	s	sampling frequency in seconds for statistical (1D and 0D) output	
avg_interval_sec	R	900	s	(time) averaging interval in seconds for 1D statistical output	
expname	C	ICOLES		expname to name the statistical output file	
ldiag_les_out	L	.FALSE.		Control for the statistical output in LES mode	
les_metric	L	.FALSE.		Switch to turn on Smagorinsky diffusion with 3D metric terms to account for topography	

Defined and used in: `src/namelist/mo_les_nml.f90`

#### 4.4.15 limarea\_nml (Scope: `limited_area=.TRUE.` in `grid_nml`)

Parameter	Type	Default	Unit	Description	Scope
itype_latbc	I	0		Type of lateral boundary nudging. Nudge from 0: the initial data, 1: IFS data analysis/forecast (if <code>initicon_nml:init_mode=4</code> , we take COSMO-DE data), 2: ICON output data (with the identical 3d grid)	
dtime_latbc	R	10800.0	s	Time difference between two consecutive boundary data.	itype_latbc $\geq 1$
nlev_latbc	I	0	s	Number of vertical levels in boundary data.	itype_latbc $\geq 1$

Parameter	Type	Default	Unit	Description	Scope
<b>latbc_filename</b>	C			Filename of boundary data input file, default: "prepiconR<nroot>B<jlev><y><m><d><h>.nc". <y>, <m>, <d>, and <h> will be automatically replaced during the run-time. In case the time span between two consecutive boundary data is less than 1 hour, one can use <min> and <sec>. These files must be located in the <b>latbc_path</b> directory.	itype.latbc $\geq$ 1
<b>latbc_path</b>	C			Absolute path to boundary data.	itype.latbc $\geq$ 1

Defined and used in: `src/namelist/mo_limarea_nml.f90`

#### 4.4.16 lnd\_nml

Parameter	Type	Default	Unit	Description	Scope
<b>nlsv_snow</b>	I	2		number of snow layers	lmulti_snow=.true.
<b>ntiles</b>	I	1		number of tiles	ntiles>1
<b>lsnowtile</b>	L	.FALSE.		.TRUE.: consider snow-covered and snow-free tiles separately	ntiles>1
<b>frlnd_thrhd</b>	R	0.05		fraction threshold for creating a land grid point	ntiles>1
<b>frlake_thrhd</b>	R	0.05		fraction threshold for creating a lake grid point	ntiles>1
<b>frsea_thrhd</b>	R	0.05		fraction threshold for creating a sea grid point	ntiles>1
<b>frlndtile_thrhd</b>	R	0.05		fraction threshold for retaining the respective tile for a grid point	ntiles>1
<b>lmelt</b>	L	.TRUE.		.TRUE. soil model with melting process	
<b>lmelt_var</b>	L	.TRUE.		.TRUE. freezing temperature dependent on water content	
<b>lana_rho_snow</b>	L	.TRUE.		.TRUE. take rho.snow-values from analysis file	init_mode=1
<b>lmulti_snow</b>	L	.FALSE.		.TRUE. for use of multi-layer snow model (default is single-layer scheme)	

Parameter	Type	Default	Unit	Description	Scope
l2lay_rho_snow	L	.FALSE.		.TRUE. predict additional snow density for upper part of the snowpack, having a maximum depth of max_toplaydepth maximum depth of uppermost snow layer	lmulti_snow = .FALSE.  lmulti_snow=.TRUE. or l2lay_rho_snow=.TRUE.
max_toplaydepth	R	0.25	m		
idiag_snowfrac	I	1		Type of snow-fraction diagnosis: 1 = based on SWE only 2-4 = more advanced experimental methods 20, 30, 40 = same as 2, 3, 4, respectively, but with artificial reduction of snow fraction in case of melting snow	
itype_lndtbl	I	3		Table values used for associating surface parameters to land-cover classes: 1 = defaults from extpar (GLC2000 and GLOBCOVER2009) 2 = Tuned version based on IFS values for globcover classes (GLOBCOVER2009 only) 3 = even more tuned operational version (GLOBCOVER2009 only) 4 = tuned version for new bare soil evaporation scheme (itype_evsl=4)	
itype_root	I	2		root density distribution: 1 = constant 2 = exponential	
itype_evsl	I	2		type of bare soil evaporation parameterization 2 = Dickinson (1984) 3 = Noilhan and Planton (1989) 4 = Resistance-based scheme by Jan-Peter Schulz	

Parameter	Type	Default	Unit	Description	Scope
itype_heatcond	I	2		type of soil heat conductivity 1 = constant soil heat conductivity 2 = moisture dependent soil heat conductivity 3 = variant of option 2 with reduced near-surface heat conductivity in the presence of plant cover	
itype_interception	I	1		type of plant interception 1 = standard scheme, effectively switched off by tiny value cwimax_ml 2 = Rain and snow interception ( <b>under development</b> )	
cwimax_ml	R	1.e-6	m	scaling parameter for maximum interception storage (almost switched off); use 5.e-4 to activate interception storage type of hydraulic lower boundary condition 1 = none 3 = ground water as lower boundary of soil column	itype_interception = 1
itype_hydbound	I	1			
lstomata	L	.TRUE.		If .TRUE., use map of minimum stomatal resistance	
l2tls	L	.TRUE.		If .FALSE., use constant value of 150 s/m. If .TRUE., forecast with 2-Time-Level integration scheme	
lseaice	L	.TRUE.		.TRUE. for use of sea-ice model	
llake	L	.TRUE.		.TRUE. for use of lake model	

Parameter	Type	Default	Unit	Description	Scope
sstice_mode	I	1		1: SST and sea ice fraction are read from the analysis and kept constant. The sea ice fraction can be modified by the seaice model. 2: SST and sea ice fraction are updated daily, based on climatological monthly means 3: SST and sea ice fraction are updated daily, based on actual monthly means 4: SST and sea ice fraction are updated daily, based on actual daily means, <b>not yet implemented</b>	iequations=3 iforcing=3
sst_td_filename	C			Filename of SST input files for time dependent SST. Default is " <path>SST_<year>-jmonth_i_<gridfile>". May contain the keyword <path> which will be substituted by model_base_dir	sstice_mode=2,3
ci_td_filename	C			Filename of sea ice fraction input files for time dependent sea ice fraction. Default is " <path>CI_<year>-<month>-<gridfile>". May contain the keyword <path> which will be substituted by model_base_dir	sstice_mode=2,3

Defined and used in: `src/namelist/mo_lnd_nwp_nml.f90`

#### 4.4.17 ls\_forcing\_nml (parameters for large-scale forcing; valid for torus geometry)

Parameter	Type	Default	Unit	Description	Scope
is_subsidence_moment	L	.FALSE.		switch for enabling LS vertical advection due to subsidence for momentum equations	is_plane_torus=.TRUE.
is_subsidence_heat	L	.FALSE.		switch for enabling LS vertical advection due to subsidence for thermal equations	is_plane_torus=.TRUE.
is_advection	L	.FALSE.		switch for enabling LS horizontal advection (currently only for thermal equations)	is_plane_torus=.TRUE.

Parameter	Type	Default	Unit	Description	Scope
is_geowind	L	.FALSE.		switch for enabling geostrophic wind	is_plane_torus=.TRUE.
is_rad_forcing	L	.FALSE.		switch for enabling radiative forcing	is_plane_torus=.TRUE. inwp_rad=.FALSE.
is_theta	L	.FALSE.		switch to indicate that the prescribed radiative forcing is for potential temperature	is_plane_torus=.TRUE. is_rad_forcing=.TRUE.

Defined and used in: `src/namelist/mo_ls_forcing_nml.f90`

#### 4.4.18 master\_model\_nml (repeated for each model)

Parameter	Type	Default	Unit	Description	Scope
model_name	C			Character string for naming this component.	
model_namelist_filename	C			File name containing the model namelists.	
model_type	I	-1		Identifies which component to run. 1=atmosphere 2=ocean 3=radiation 99=dummy_model	
model_min_rank	I	0		Start MPI rank for this model.	
model_max_rank	I	-1		End MPI rank for this model.	
model_inc_rank	I	1		Stride of MPI ranks.	

#### 4.4.19 master\_nml

Parameter	Type	Default	Unit	Description	Scope
lrestart	L	.FALSE.		If .TRUE.: Current experiment is started from a restart.	

Parameter	Type	Default	Unit	Description	Scope
<b>model_base_dir</b>	C	, ,		General path which may be used in file names of other name lists: If a file name contains the keyword "<path>", then this <b>model_base_dir</b> will be substituted.	

#### 4.4.20 meteogram\_output\_nml

Nearest neighbour 'interpolation' is used for all variables.

Parameter	Type	Default	Unit	Description	Scope
lmeteogram_enabled	L(n_dom)	.FALSE.		Flag. True, if meteogram of output variables is desired.	
zprefix	C(n_dom)	"METEOGRAM"		string with file name prefix for output file	
ldistributed	L(n_dom)	.TRUE.		Flag. Separate files for each PE.	
loutput_tiles	L	.FALSE.		Write tile-specific output for some selected surface/soil fields	
n0_mtgrm	I(n_dom)	0		initial time step for meteogram output.	
ninc_mtgrm	I(n_dom)	1		output interval (in time steps)	
stationlist_tot		53.633, 9.983, 'Ham- burg' " "		list of meteogram stations (triples with lat, lon, name string)	
var_list	C(:)			Positive-list of variables (optional). Only variables contained in this list are included in the meteogram. If the default list is not changed by user input, then all available variables are added to the meteogram	

Defined and used in: **src/namelist/mo\_mtgrm\_nml.f90**

## 4.4.21 nonhydrostatic\_nml (relevant if run\_nml:iequations=3)

Parameter	Type	Default	Unit	Description	Scope
<b>itime_scheme</b>	I	4		Options for predictor-corrector time-stepping scheme: 4: Contravariant vertical velocity is computed in the predictor step only, velocity tendencies are computed in the corrector step only (most efficient option) 5: Contravariant vertical velocity is computed in both substeps (beneficial for numerical stability in very-high resolution setups with extremely steep slopes, otherwise no significant impact) 6: As 5, but velocity tendencies are also computed in both substeps (no apparent benefit, but more expensive) Type of Rayleigh damping 1: CLASSICAL (requires velocity reference state.) 2: Klemp (2008) type Rayleigh damping coefficient $1/\tau_0$ (Klemp, Dudhia, Hassiotis: MWR136, pp.3987-4004); higher values are recommended for R2B6 or finer resolution Height at which Rayleigh damping of vertical wind starts (needs to be adjusted to model top height; the damping layer should have a depth of at least 20 km when the model top is above the stratopause) Height above which moist physics and advection of cloud and precipitation variables are turned off	iequations=3
<b>rayleigh_type</b>	I	2			
<b>rayleigh_coeff</b>	R(n_dom)	0.05 for i=1			
<b>damp_height</b>	R(n_dom)	45000 for i=1	m		
<b>htop_moist_proc</b>	R	22500.0	m		



Parameter	Type	Default	Unit	Description	Scope
hbot_qvsubstep	R	22500.0	m	Height above which QV is advected with substepping scheme (must be at least as large as htop_moist_proc)	ihadv_tracer=22, 32, 42 or 52
vwind_offctr	R	0.15		Off-centering in vertical wind solver. Higher values may be needed for R2B5 or coarser grids when the model top is above 50 km.	
rhotheta_offctr	R	-0.1		Off-centering of density and potential temperature at interface level (may be set to 0.0 for R2B6 or finer grids)	
veladv_offctr	R	0.25		Off-centering of velocity advection in corrector step	
ivctype	I	2		Type of vertical coordinate: 1: Gal-Chen hybrid	
ndyn_substeps	I	5		2: SLEVE (uses sleve.nml) number of dynamics substeps per fast-physics / transport step	
lhdiff_rcf	L	.TRUE.		.TRUE.: Compute diffusion only at advection time steps (in this case, divergence damping is applied in the dynamical core)	
lextra_diffu	L	.TRUE.		.TRUE.: Apply additional momentum diffusion at grid points close to the stability limit for vertical advection (becomes effective extremely rarely in practice; this is mostly an emergency fix for pathologic cases with very large orographic gravity waves)	
divdamp_fac	R	0.0025		Scaling factor for divergence damping	lhdiff_rcf = .TRUE.

Parameter	Type	Default	Unit	Description	Scope
divdamp_order	I	4		Order of divergence damping: 2 = second-order divergence damping 4 = fourth-order divergence damping 24 = combined second-order and fourth-order divergence damping and enhanced vertical wind off-centering during the initial spinup phase (does not allow checkpointing/restarting earlier than 2.5 hours of integration)	lhdiff_ref = .TRUE.
divdamp_type	I	3		Type of divergence damping: 2 = divergence damping acting on 2D divergence 3 = divergence damping acting on 3D divergence 32 = combination of 3D div. damping in the troposphere with transition to 2D div. damping in the stratosphere	lhdiff_ref = .TRUE.
divdamp_trans_start	R	12500.		Lower bound of transition zone between 2D and 3D divergence damping	divdamp_type = 32
divdamp_trans_end	R	17500.		Upper bound of transition zone between 2D and 3D divergence damping	divdamp_type = 32
nest_substeps	I	2		Number of dynamics substeps for the child patches. <b>DO NOT CHANGE!!! The code will not work correctly with other values</b>	
l_masscorr_nest	L	.FALSE.		.TRUE.: Apply mass conservation correction also in nested domain	
iadv_rhotheta	I	2		Advection method for rho and rhotheta: 1: simple second-order upwind-biased scheme 2: 2nd order Miura horizontal 3: 3rd order Miura horizontal (not recommended)	

Parameter	Type	Default	Unit	Description	Scope
igradp_method	I	3		Discretization of horizontal pressure gradient: 1: conventional discretization with metric correction term 2: Taylor-expansion-based reconstruction of pressure (advantageous at very high resolution) 3: Similar discretization as option 2, but uses hydrostatic approximation for downward extrapolation over steep slopes 4: Cubic/quadratic polynomial interpolation for pressure reconstruction 5: Same as 4, but hydrostatic approximation for downward extrapolation over steep slopes .TRUE.: Compute Smagorinsky temperature diffusion truly horizontally over steep slopes Slope threshold above which truly horizontal temperature diffusion is activated	hdiff_order=3/5 .AND. lhdiff_temp = .true. hdiff_order=3/5 .AND. lhdiff_temp=.true. .AND. lzdifft=.true. hdiff_order=3/5 .AND. lhdiff_temp=.true. .AND. lzdifft=.true.
lzdifft	L	.TRUE.			
thslp_zdifft	R	0.025			
thhgtd_zdifft	R	200	m	Threshold of height difference between neighboring grid points above which truly horizontal temperature diffusion is activated (alternative criterion to thslp_zdifft)	
exner_expol	R	1./3.		Temporal extrapolation (fraction of dt) of Exner function for computation of horizontal pressure gradient. This damps horizontally propagating sound waves. For R2B5 or coarser grids, values between 1/2 and 2/3 are recommended.	
l_open_abc	L	.FALSE.		.TRUE.: Use open upper boundary condition (rather than w=0) to allow vertical motions related to diabatic heating to extend beyond the model top	

Defined and used in: src/namelist/mo\_nonhydrostatic\_nml.f90

#### 4.4.22 nwp\_phy\_nml

The switches for the physics schemes and the time steps can be set for each model domain individually. If only one value is specified, it is copied to all child domains, implying that the same set of parameterizations and time steps is used in all domains. If the number of values given in the namelist is larger than 1 but less than the number of model domains, then the settings from the highest domain ID are used for the remaining model domains. If the time steps are not an integer multiple of the advective time step (dtime), then the time step of the respective physics parameterization is automatically rounded to the next higher integer multiple of the advective time step.

Parameter	Type	Default	Unit	Description	Scope
<b>inwp_gscp</b>	I (max_dom)	1		cloud microphysics and precipitation 0: none 1: hydc1 (COSMO-EU microphysics, 2-cat ice: cloud ice, snow) 2: hydc1_gr (COSMO-DE microphysics, 3-cat ice: cloud ice, snow, graupel) 3: as 1, but with improved ice nucleation scheme by C. Koehler 4: Two-moment microphysics by A. Seifert 9: Kessler scheme	run_nml:iforcing = inwp
qi0	R	0.0	kg/kg	cloud ice threshold for autoconversion	inwp_gscp=1
qc0	R	0.0	kg/kg	cloud water threshold for autoconversion	inwp_gscp=1
mu_rain	R	0.0		shape parameter in gamma distribution for rain	inwp_gscp>0
mu_snow	R	0.0		shape parameter in gamma distribution for snow	inwp_gscp>0
icpl_aero_gscp	I	0		0: off 1: simple coupling between autoconversion and Tegen aerosol climatology; requires irad_aero=6 More advanced options are in preparation	currently only for inwp_gscp = 1
<b>inwp_convection</b>	I (max_dom)	1		convection 0: none 1: Tiedtke/Bechtold convection .TRUE.: use shallow convection only	run_nml:iforcing = inwp
lshallowconv_only	L (max_dom)	.FALSE.			inwp_convection = 1

Parameter	Type	Default	Unit	Description	Scope
icapdcycl	I	0		Type of CAPE correction to improve diurnal cycle for convection: 0 = none (IFS default prior to autumn 2013) 1 = intermediate testing option 2 = corrects over land and water now operational at ECMWF 3 = correction over land as in 2 restricted to the tropics, no correction over water (this choice optimizes the NWP skill scores)	inwp_convection = 1
icpl_aero_conv	I	0		0: off 1: simple coupling between autoconversion and Tegen aerosol climatology; requires irad_aero=6	
iprog_aero	I	0		0: off 1: simple prognostic aerosol scheme, based on 2D aerosol optical depth fields of Tegen climatology; requires irad_aero=6	
icpl_o3_tp	I	1		0: off 1: simple coupling between the ozone mixing ratio and the thermal tropopause, restricted to the extratropics	irad_o3 = 7 or 9
<b>inwp_cldcover</b>	I (max_dom)	1		cloud cover scheme for radiation 0: no clouds (only QV) 1: diagnostic cloud cover (by Martin Koehler) 2: prognostic total water variance (not yet started) 3: clouds from COSMO SGS cloud scheme 4: clouds as in turbulence (turbdiff) 5: grid scale clouds	run_nml:iforcing = inwp
<b>inwp_radiation</b>	I (max_dom)	1		radiation 0: none 1: RRTM radiation 2: Ritter-Geleyn radiation 3: PSRAD radiation	run_nml:iforcing = inwp

Parameter	Type	Default	Unit	Description	Scope
<b>inwp_satad</b>	I	1		saturation adjustment 0: none 1: saturation adjustment at constant density	run_nml:iforcing = inwp
<b>inwp_turb</b>	I (max_ dom)	1		vertical diffusion and transfer 0: none 1: COSMO diffusion and transfer 2: GME turbulence scheme 3: EDMF-DUALM (work in progress) 5: Classical Smagorinsky diffusion subgrid scale orographic drag 0: none 1: Lott and Miller scheme (COSMO) non-orographic gravity wave drag 0: none 1: Orr-Ern-Bechtold-scheme (IFS) surface scheme 0: none 1: TERRA	run_nml:iforcing = inwp
<b>inwp_sso</b>	I (max_ dom)	1		wind speed at which extra Rayleigh friction starts	run_nml:iforcing = inwp
<b>inwp_gwd</b>	I (max_ dom)	1		minimum e-folding time of Rayleigh friction (effective for $u > \text{ustart\_raylfric} + 90$ m/s)	run_nml:iforcing = inwp
<b>inwp_surface</b>	I (max_ dom)	1		.TRUE.: take into account atmosphere above model top for radiation computation	run_nml:iforcing = inwp
ustart_raylfric	R	160.0	m/s	Type of roughness length data used for turbulence scheme:	inwp_gwd > 0
efdt_min_raylfric	R	10800.	s	1 = land-cover-related roughness including contribution from sub-scale orography	inwp_gwd > 0
latm_above_top	L (max_ dom)	.FALSE.		2 = land-cover-related roughness only	inwp_radiation > 0
itype_z0	I	2		time interval of convection call	inwp_turb > 0
<b>dt_conv</b>	R (max_ dom)	600.	s	currently each subdomain has the same value	run_nml:iforcing = inwp
<b>dt_rad</b>	R (max_ dom)	1800.	s	time interval of radiation call	run_nml:iforcing = inwp
				currently each subdomain has the same value	run_nml:iforcing = inwp

Parameter	Type	Default	Unit	Description	Scope
<b>dt_sso</b>	R (max_dom)	1200.	s	time interval of sso call	run_nml:iforcing = inwp
<b>dt_gwd</b>	R (max_dom)	1200.	s	time interval of gwd call	run_nml:iforcing = inwp
lrtnm_filename	C(:)	“rtrtmg-lw.nc”		currently each subdomain has the same value NetCDF file containing longwave absorption coefficients and other data for RRTMG_LW k-distribution model.	
cldopt_filename	C(:)	“ECHAM6_CldOpt Props.nc”		NetCDF file with RRTM Cloud Optical Properties for ECHAM6.	

Defined and used in: `src/namelist/mo_nwp_phy_nml.f90`

#### 4.4.23 nwp\_tuning\_nml

Please note: These tuning parameters are NOT domain specific.

Parameter	Type	Default	Unit	Description	Scope
<b>SSO</b> (Lott and Miller)					
tune_gkwake	R	1.5		low level wake drag constant	run_nml:iforcing = inwp
tune_gkdrag	R	0.075		gravity wave drag constant	run_nml:iforcing = inwp
<b>GWD</b> (Warner McIntyre)					
tune_gfluxlaun	R	2.50e-3		total launch momentum flux in each azimuth (rho_o x F_o)	run_nml:iforcing = inwp
<b>Grid scale microphysics</b> (one moment)					
tune_zceff_min	R	0.075		Minimum value for sticking efficiency	run_nml:iforcing = inwp
tune_v0snow	R	25.0		factor in the terminal velocity for snow	run_nml:iforcing = inwp

Parameter	Type	Default	Unit	Description	Scope
tune_zvz0i	R	1.25	m/s	Terminal fall velocity of ice	run_nml:iforcing = inwp
<b>Convection scheme</b>					
tune_entrorg	R	1.85e-3	1/m	Entrainment parameter valid for dx=20 km (depends on model resolution)	run_nml:iforcing = inwp
tune_capdcfac_et	R	0		Fraction of CAPE diurnal cycle correction applied in the extratropics	icapdcycl = 3
tune_rhebc_land	R	0.75		RH threshold for onset of evaporation below cloud base over land	run_nml:iforcing = inwp
tune_rhebc_land_trop	R	0.70		RH threshold for onset of evaporation below cloud base over land in the tropics	run_nml:iforcing = inwp
tune_rhebc_ocean	R	0.85		RH threshold for onset of evaporation below cloud base over sea	run_nml:iforcing = inwp
tune_rhebc_ocean_trop	R	0.80		RH threshold for onset of evaporation below cloud base over sea in the tropics	run_nml:iforcing = inwp
tune_rcucov	R	0.05		Convective area fraction used for computing evaporation below cloud base	run_nml:iforcing = inwp
tune_rcucov_trop	R	0.05		Convective area fraction used for computing evaporation below cloud base in the tropics	run_nml:iforcing = inwp
tune_textc	R	0.125	K	Excess value for temperature used in test parcel ascent	run_nml:iforcing = inwp
tune_qexc	R	0.0125		Excess fraction of grid-scale QV used in test parcel ascent	run_nml:iforcing = inwp
<b>Misc</b>					
itune_albedo	I	0		MODIS albedo tuning 0: None 1: dimmed sahara	run_nml:iforcing = inwp albedo_type=2
tune_minsnowfrac	R	0.125		Minimum value to which the snow cover fraction is artificially reduced in case of melting show	lnd_nml:idiag_snowfrac = 20/30/40
<b>IAU</b>					



Parameter	Type	Default	Unit	Description	Scope
max_freshsnow_inc	R	0.025		Maximum allowed freshsnow increment per analysis cycle (positive or negative)	init_mode=5 (MODE_IAU)

Defined and used in: `src/namelist/mo_nwp_tuning_nml.f90`

#### 4.4.24 output\_nml (relevant if `run_nml/output='nml'`)

Please note: There may be several instances of `output_nml` in the namelist file, every one defining a list of variables with separate attributes for output.

Parameter	Type	Default	Unit	Description	Scope
<b>dom</b>	I(:)	-1		Array of domains for which this name-list is used. If not specified (or specified as -1 as the first array member), this name-list will be used for all domains. Attention: Depending on the setting of the parameter <code>l_output_phys_patch</code> these are either logical or physical domain numbers! Defines the length of a file in terms of an ISO-8601 duration string. An example for this time stamp format is given below. This namelist parameter can be set instead of <b>steps_per_file</b> .	
<b>file_interval</b>	C	‘ ‘			
<b>filename_format</b>	C	see description.		Output filename format. Includes keywords <code>path</code> , <code>output_filename</code> , <code>physdom</code> , etc. (see below). Default is <code>&lt;output_filename&gt;_DOM&lt;physdom&gt;_&lt;levtype&gt;_&lt;jfile&gt;</code>	
<b>filename_extn</b>	C	”default”		User-specified filename extension (empty string also possible). If this namelist parameter is chosen as ”default”, then we have ”.nc” for NetCDF output files, and ”.grb” for GRIB1/2.	

Parameter	Type	Default	Unit	Description	Scope
filetype	I	4		One of CDI's FILETYPE_XXX constants. Possible values: 2=FILETYPE_GRB2, 4=FILETYPE_NC2, 5=FILETYPE_NC4 Model level indices (optional). Allowed is a comma- (or semicolon-) separated list of integers, and of integer ranges like "10...20". One may also use the keyword "nlev" to denote the maximum integer (or, equivalently, "n" or "N"). Furthermore, arithmetic expressions like "(nlev - 2)" are possible. Basic example: <b>m_levels = "1,3,5...10,20...(nlev-2)"</b>	
m_levels	C	None			
h_levels	R(:)	None	m	height levels	
p_levels	R(:)	None	Pa	pressure levels	
i_levels	R(:)	None	K	isentropic levels	
ml_varlist	C(:)	None		Name of model level fields to be output.	
hl_varlist	C(:)	None		Name of height level fields to be output.	
pl_varlist	C(:)	None		Name of pressure level fields to be output.	
il_varlist	C(:)	None		Name of isentropic level fields to be output.	
include_last	L	.TRUE.		Flag whether to include the last time step	
mode	I	2		1 = forecast mode, 2 = climate mode In climate mode the time axis of the output file is set to TAXIS_ABSOLUTE. In forecast mode it is set to TAXIS_RELATIVE. Till now the forecast mode only works if the output is at multiples of 1 hour	

Parameter	Type	Default	Unit	Description	Scope
<code>taxis_tunit</code>	I	2		Time unit of the TAXIS_RELATIVE time axis. 1 = TUNIT_SECOND 2 = TUNIT_MINUTE 5 = TUNIT_HOUR 9 = TUNIT_DAY For a complete list of possible values see <code>cdilib.c</code>	mode=1
<code>output_bounds</code>	R( $k*3$ )	None		Post-processing times: start, end, increment. We choose the advection time step matching or following the requested output time, therefore we require <code>output_bounds(3) &gt; dttime</code> . Multiple triples are possible in order to define multiple starts/ends/intervals. See namelist parameters <code>output_start</code> , <code>output_end</code> , <code>output_interval</code> for an alternative specification of output events.	
<code>output_time_unit</code>	I	1		Units of output bounds specification. 1 = second 2 = minute 3 = hour 4 = day 5 = month 6 = year	
<code>output_filename</code>	C	None		Output filename prefix (which may include path). Domain number, level type, file number and extension will be added, according to the format given in namelist parameter “ <code>filename_format</code> ”.	
<code>output_grid</code>	L	.FALSE.		Flag whether grid information is added to output.	

Parameter	Type	Default	Unit	Description	Scope
<b>output_start</b>	C(:)	‘ ‘		ISO8601 time stamp for begin of output. An example for this time stamp format is given below. More than one value is possible in order to define multiple start/end/interval triples. See namelist parameter <b>output_bounds</b> for an alternative specification of output events.	
<b>output_end</b>	C(:)	‘ ‘		ISO8601 time stamp for end of output. An example for this time stamp format is given below. More than one value is possible in order to define multiple start/end/interval triples. See namelist parameter <b>output_bounds</b> for an alternative specification of output events.	
<b>output_interval</b>	C(:)	‘ ‘		ISO8601 time stamp for repeating output intervals. We choose the advection time step matching or following the requested output time, therefore we require <b>output_bounds(3) &gt; dtime</b> . An example for this time stamp format is given below. More than one value is possible in order to define multiple start/end/interval triples. See namelist parameter <b>output_bounds</b> for an alternative specification of output events.	
<b>pe_placement_il</b>	I(:)	-1		Advanced output option: Explicit assignment of output MPI ranks to the isentropic level output file. At most <b>stream_partitions_il</b> different ranks can be specified. See namelist parameter <b>pe_placement_ml</b> for further details.	
<b>pe_placement_hl</b>	I(:)	-1		Advanced output option: Explicit assignment of output MPI ranks to the height level output file. At most <b>stream_partitions_hl</b> different ranks can be specified. See namelist parameter <b>pe_placement_ml</b> for further details.	

Parameter	Type	Default	Unit	Description	Scope
pe_placement_ml	I(:)	-1		Advanced output option: Explicit assignment of output MPI ranks to the model level output file. At most <code>stream_partitions_ml</code> different ranks can be specified, out of the following list: 0 ... ( <code>num_io_procs</code> - 1). If this namelist parameters is not provided, then the output ranks are chosen in a Round-Robin fashion among those ranks that are not occupied by explicitly placed output files.	
pe_placement_pl	I(:)	-1		Advanced output option: Explicit assignment of output MPI ranks to the pressure level output file. At most <code>stream_partitions_pl</code> different ranks can be specified. See namelist parameter <code>pe_placement_ml</code> for further details.	
ready_file	C	'default'		A <i>ready_file</i> is a technique for handling dependencies between the NWP processes. The completion of the write process is signalled by creating a small file with name <code>ready_file</code> . Different <code>output_nml</code> 's may be joined together to form a single ready file event. The setting of <code>ready_file</code> = "default" does not create a ready file. The ready file name may contain string tokens <code>&lt;path&gt;</code> , <code>&lt;datetime&gt;</code> , <code>&lt;ddhhmmss&gt;</code> which are substituted as described for the namelist parameter <code>filename_format</code> .	
reg_def_mode	I	0		Specify if the "delta" value prescribes an interval size or the total *number* of intervals: 0: switch automatically between increment and no. of grid points, 1: <code>reg_lon/lat_def(2)</code> specifies increment, 2: <code>reg_lon/lat_def(2)</code> specifies no. of grid points.	remap=1

Parameter	Type	Default	Unit	Description	Scope
<b>remap</b>	I	0		interpolate horizontally 0: none	
north_pole	R(2)	0,90		1: to regular lat-lon grid definition of north pole for rotated lon-lat grids ([ <b>longitude</b> , <b>latitude</b> ]).	
<b>reg_lat_def</b>	R(3)	None		start, increment, end latitude in degrees. Alternatively, the user may set the number of grid points instead of an increment. Details for the setting of regular grids is given below together with an example.	remap=1
<b>reg_lon_def</b>	R(3)	None		The regular grid points are specified by three values: start, increment, end given in degrees. Alternatively, the user may set the number of grid points instead of an increment. Details for the setting of regular grids is given below together with an example.	remap=1
<b>steps_per_file</b>	I	-1		Max number of output steps in one output file. If this number is reached, a new output file will be opened.	
steps_per_file_inclfirst	L	see descr.		Defines if first step is counted wrt. <b>steps_per_file</b> files count. The default is <b>.FALSE.</b> for GRIB2 output, and <b>.TRUE.</b> otherwise.	
stream_partitions_hl	I	1		Splits height level output of this namelist into several concurrent alternating files. See namelist parameter <b>stream_partitions_m1</b> for details.	
stream_partitions_il	I	1		Splits isentropic level output of this namelist into several concurrent alternating files. See namelist parameter <b>stream_partitions_m1</b> for details.	

Parameter	Type	Default	Unit	Description	Scope
stream_partitions_ml	I	1		Splits model level output of this namelist into several concurrent alternating files. The output is split into $N$ files, where the start date of part $i$ gets an offset of $(i - 1) * \text{output\_interval}$ . The output interval is then replaced by $N * \text{output\_interval}$ , the <code>include_last</code> flag is set to <code>.FALSE.</code> , the <code>steps_per_file_inclfirst</code> flag is set to <code>.FALSE.</code> , and the <code>steps_per_file</code> counter is set to 1.	
stream_partitions_pl	I	1		Splits pressure level output of this namelist into several concurrent alternating files. See namelist parameter <code>stream_partitions_ml</code> for details.	
rbf_scale	R	-1.		Explicit setting of RBF shape parameter for interpolated lon-lat output. This namelist parameter is only active in combination with <code>interpol_nml:rbf_scale_mode_ll=3</code> .	<code>interpol_nml:rbf_scale_mode_ll=3</code>

Defined and used in: `src/io/shared/mo_name_list_output_init.f90`

**Interpolation onto regular grids:** Horizontal interpolation onto regular grids is possible through the namelist setting `remap=1`, where the mesh is defined by the parameters

- `reg_lon_def`: mesh latitudes in degrees,
- `reg_lat_def`: mesh longitudes in degrees,
- `north_pole`: definition of north pole for rotated lon-lat grids.

The regular grid points in `reg_lon_def`, `reg_lat_def` are each specified by three values, given in degrees: *start*, *increment*, *end*. The mesh then contains all grid points  $start + k * \text{increment} \leq end$ , where  $k$  is an integer. Instead of defining an increment it is also possible to prescribe the number of grid points.

- Setting the namelist parameter `reg_def_mode=0`: Switch automatically from increment specification to no. of grid points, when the `reg_lon/lat_def(2)` value is larger than 5.0.

- 1: `reg_lon/lat_def(2)` specifies increment
- 2: `reg_lon/lat_def(2)` specifies no. of grid points

For longitude values the last grid point is omitted if the end point matches the start point, e.g. for 0 and 360 degrees.

### Examples

local grid with 0.5 degree increment:

```
reg_lon_def = -30., 0.5, 30.
reg_lat_def = 90., -0.5, -90.
```

global grid with 720x361 grid points:

```
reg_lon_def = 0., 720, 360.
reg_lat_def = -90., 360, 90.
```

**Time stamp format:** The namelist parameters `output_start`, `output_end`, `output_interval` allow the specification of time stamps according to ISO 8601. The general format for time stamps is `YYYY-MM-DDThh:mm:ss` where Y: year, M: month, D: day for dates, and hh: hour, mm: minute, ss: second for time strings. The general format for durations is `PnYnMnDnHnMnS`. See, for example, [http://en.wikipedia.org/wiki/ISO\\_8601](http://en.wikipedia.org/wiki/ISO_8601) for details and further specifications.

**NOTE:** as the `mtime` library underlying the output driver currently has some restrictions concerning the specification of durations:

1. Any number `n` in `PnYnMnDnHnMnS` must have two digits. For instance use `"PT06H"` instead of `"PT6H"`
2. In a duration string `PnyearYnmonMndayDtnhrHnminMnsecS` the numbers `nxyz` must not pass the carry over number to the next larger time unit: `0j=nmonj=12, 0j=nhrj=23, 0j=nminj=59, 0j=nsecj=59.999`. For instance use `"P01D"` instead of `"PT24H"`, or `"PT01M"` instead of `"PT60S"`.

Soon the formatting problem will be resolved and the valid number ranges will be enlarged. (2013-12-16).

### Examples

date and time representation (`output_start`, `output_end`)  
duration (`output_interval`)

```
2013-10-27T13:41:00Z
POODT06H00M00S
```



### Variable Groups

**Keyword "group":** Using the "group:" keyword for the namelist parameters `ml_varlist`, `hl_varlist`, `pl_varlist`, sets of common variables can be added to the output:

<code>group:all</code>	output of all variables (caution: do not combine with <u>mixed</u> vertical interpolation)
<code>group:atmo_ml_vars</code>	basic atmospheric variables on model levels
<code>group:atmo_pl_vars</code>	same set as <code>atmo_ml_vars</code> , but except pres
<code>group:atmo_zl_vars</code>	same set as <code>atmo_ml_vars</code> , but expect height
<code>group:nh_prog_vars</code>	additional prognostic variables of the nonhydrostatic model
<code>group:atmo_derived_vars</code>	derived atmospheric variables
<code>group:rad_vars</code>	
<code>group:precip_vars</code>	
<code>group:cloud_diag</code>	
<code>group:pbl_vars</code>	
<code>group:phys_tendencies</code>	
<code>group:land_vars</code>	
<code>group:snow_vars</code>	snow variables
<code>group:multisnow_vars</code>	multi-layer snow variables
<code>group:additional_precip_vars</code>	
<code>group:dwd_fg_atm_vars</code>	DWD first guess fields (atmosphere)
<code>group:dwd_fg_sfc_vars</code>	DWD first guess fields (surface/soil)
<code>group:ART_AERO_VOLC</code>	ART volcanic ash fields
<code>group:ART_AERO_RADIO</code>	ART radioactive tracer fields
<code>group:ART_AERO_DUST</code>	ART mineral dust aerosol fields
<code>group:ART_AERO_SEAS</code>	ART sea salt aerosol fields
<code>group:prog_timemean</code>	time mean output: temp, u, v, rho
<code>group:tracer_timemean</code>	time mean output: qv, qc, qi
<code>group:echam_timemean</code>	time mean output: most echam surface variables
<code>group:atmo_timemean</code>	time mean variables from <code>prog_timemean</code> , <code>tracer_timemean</code> , <code>echam_timemean</code>

**Keyword "tiles:":** The "tiles:" keyword allows to add all tiles of a specific variable to the output, without the need to specify all tile fields separately. E.g. "tiles:t\_g" (read: "tiles of t\_g") automatically adds all t\_g.t.X fields to the output. Here, X is a placeholder for the tile number. Make sure to specify the name of the aggregated variable rather than the name of the corresponding tile container (i.e. in the given example it must be t\_g, and not t\_g.t!).

**Note:**

There exists a special syntax which allows to remove variables from the output list, e.g. if these undesired variables were contained in a previously selected group.

Typing "-[varname]" (for example "-temp") removes the variable from the union set of group variables and other selected variables. Note that typos are not detected but that the corresponding variable is simply not removed!

**Keyword substitution in output filename (filename\_format):**

path	substituted by model_base_dir
output_filename	substituted by output_filename
physdom	substituted by physical patch ID
levtype	substituted by level type "ML", "PL", "HL", "IL"
levtype_l	like levtype, but in lower case
jfile	substituted by output file counter
datetime	substituted by ISO-8601 date-time stamp in format YYYY-MM-DDThh:mm:ss.sssZ
datetime2	substituted by ISO-8601 date-time stamp in format YYYYMMDDThhmmssZ
datetime3	substituted by ISO-8601 date-time stamp in format YYYYMMDDThhmmss.sssZ
ddhhmmss	substituted by <i>relative</i> day-hour-minute-second string
hhmmss	substituted by <i>relative</i> hour-minute-second string
npartitions	If namelist is split into concurrent files: number of stream partitions.
ifile.partition	If namelist is split into concurrent files: stream partition index of this file.
total_index	If namelist is split into concurrent files: substituted by the file counter (like in jfile), which an "unsplit" namelist would have produced

**4.4.25 parallel\_nml**

Parameter	Type	Default	Unit	Description	Scope
<b>nproma</b>	I	1		chunk length	
n_ghost_rows	I	1		number of halo cell rows	
division_method	I	1		method of domain decomposition 0: read in from file 1: use built-in geometric subdivision	
division_file_name	C			Name of division file	division_method = 0
ldiv_phys_dom	L	.TRUE.		.TRUE.: split into physical domains before computing domain decomposition (in case of merged domains) (This reduces load imbalance; turning off this option is not recommended except for very small processor numbers)	division_method = 1
p_test_run	L	.FALSE.		.TRUE. means verification run for MPI parallelization (PE 0 processes full domain)	
l_test_openmp	L	.FALSE.		if .TRUE. is combined with p_test_run=.TRUE. and OpenMP parallelization, the test PE gets only 1 thread in order to verify the OpenMP parallelization	
l_log_checks	L	.FALSE.		if .TRUE. messages are generated during each synchronization step (use for debugging only)	
l_fast_sum	L	.FALSE.		if .TRUE., use fast (not processor-configuration-invariant) global summation	
use_dycore_barrier	L	.FALSE.		if .TRUE., set an MPI barrier at the beginning of the nonhydrostatic solver (do not use for production runs!)	
itype_exch_barrier	I	0		1: set an MPI barrier at the beginning of each MPI exchange call 2: set an MPI barrier after each MPI WAIT call 3: 1+2 (do not use for production runs!)	

Parameter	Type	Default	Unit	Description	Scope
<code>iorder_sendrecv</code>	I	1		Sequence of send/receive calls: 1 = irecv/send 2 = isend/rcv 3 = isend/irecv	
<code>itype_comm</code>	I	1		1: use local memory for exchange buffers 3: asynchronous halo communication for dynamical core (currently deactivated)	
<code>num_io_procs</code>	I	0		Number of I/O processors (running exclusively for doing I/O)	
<code>num_restart_procs</code>	I	0		Number of restart processors (running exclusively for doing restart)	
<code>num_prefetch_proc</code>	I	0		Number of processors for prefetching of boundary data asynchronously for a limited area run (running exclusively for reading Input boundary data. Maximum no of processors used for it is limited to 1).	$itype\_latbc \geq 1$
<code>pio_type</code>	I	1		Type of parallel I/O. Only used if number of I/O processors greater than number of domains. Experimental!	
<code>use_icon_comm</code>	L	.FALSE.		Enable the use of MPI bulk communication through the <code>icon_comm_lib</code>	
<code>icon_comm_debug</code>	L	.FALSE.		Enable debug mode for the <code>icon_comm_lib</code>	
<code>max_send_recv_buffer_size</code>	I	131072		Size of the send/receive buffers for the <code>icon_comm_lib</code> .	
<code>use_dp_mpi2io</code>	L	.FALSE.		Enable this flag if output fields shall be gathered by the output processes in DOUBLE PRECISION.	
<code>restart_chunk_size</code>	I	1		( <i>Advanced namelist parameter:</i> ) Number of levels to be buffered by the asynchronous restart process. The (asynchronous) restart is capable of writing and communicating more than one 2D slice at once.	

Defined and used in: `src/namelist/mo_parallel_nml.f90`

#### 4.4.26 `psrad_nml`

Parameter	Type	Default	Unit	Description	Scope
<code>lradforcing</code>	<code>L(2)</code>	<code>.FALSE.</code>		switch for diagnostics of aerosol forcing in the solar spectral range ( <code>lradforcing(1)</code> ) and the thermal spectral range ( <code>lradforcing(2)</code> ).	
<code>lw_gpts_ts</code>	<code>I</code>	<code>1</code>		number of g-points in Monte-Carlo spectral integration for thermal radiation, see <code>lw_spec_samp</code>	
<code>lw_spec_samp</code>	<code>I</code>	<code>1</code>		sampling of spectral bands in radiation calculation for thermal radiation <code>lw_spec_samp = 1</code> : standard broad band sampling <code>lw_spec_samp = 2</code> : Monte-Carlo spectral integration ( <code>MSCI</code> ); <code>lw_gpts_ts</code> randomly chosen g-points per column and radiation call <code>lw_spec_samp = 3</code> : choose g-points not completely randomly in order to reduce errors in the surface radiative fluxes	
<code>rad_perm</code>	<code>I</code>	<code>0</code>		integer number that influences the perturbation of the random seed from column to column	
<code>sw_gpts_ts</code>	<code>I</code>	<code>1</code>		number of g-points in Monte-Carlo spectral integration for solar radiation, see <code>sw_spec_samp</code>	

Parameter	Type	Default	Unit	Description	Scope
sw_spec_samp	I	1		sampling of spectral bands in radiation calculation for solar radiation sw_spec_samp = 1: standard broad band sampling sw_spec_samp = 2: Monte-Carlo spectral integration (MSCI); lw_gpts.ts randomly chosen g-points per column and radiation call sw_spec_samp = 3: choose g-points not completely randomly in order to reduce errors in the surface radiative fluxes	

Defined and used in: src/echam\_phy\_psrاد/mo\_psrاد\_radiation.f90

#### 4.4.27 psrad\_orbit\_nml

Parameter	Type	Default	Unit	Description	Scope
cecc	R	0.016715		eccentricity of earth's orbit	
coblд	R	23.44100		obliquity of earth in degrees	
l_orbsop87	L	.TRUE.		switch on (.TRUE.) the (real) observed orbit of the earth (not idealized) or switch it off (.FALSE.). In the latter case, a Kepler orbit is used.	
l_sph_symm_irr	L	.FALSE.		switch on (.TRUE.) a spherically symmetric irradiation from all sides or use an irradiation by a point source like the sun (.FALSE.).	

Defined and used in: src/namelistс/mo\_psrاد\_radiation.f90

#### 4.4.28 radiation\_nml

Parameter	Type	Default	Unit	Description	Scope
ldiur	L	.TRUE.		switch for solar irradiation: .TRUE.:diurnal cycle, .FALSE.:zonally averaged irradiation	
nmonth	I	0		0: Earth circles on orbit 1-12: Earth orbit position fixed for specified month	
lyr-perp	L	.FALSE.		.FALSE.: transient Earth orbit following VSOP87 .TRUE.: Earth orbit of year yr-perp of the VSOP87 orbit is perpetuated	
yr-perp	L	-99999		year used for lyr-perp = .TRUE. Insolation scheme	
isolrad	I	0		0: Use original SRTM insolation. 1: Use insolation from external file containing the spectrally resolved insolation (monthly means) 2: Use preindustrial insolation as in CMIP5 (average from 1844–1856) 3: Use insolation for AMIP-type CMIP5 simulation (average from 1979–1988) 4: Use insolation for RCE-type simulation with $\cos(\text{zenith angle}) = \pi/4$ (with PSRAD: use “4” if the diurnal cycle is switched on) 5: Use insolation for RCE-type simulation with PSRAD if the diurnal cycle is switched off.	

Parameter	Type	Default	Unit	Description	Scope
<b>izenith</b>	I	4		Choice of zenith angle formula for the radiative transfer computation. 0: Sun in zenith everywhere 1: Zenith angle depends only on latitude 2: Zenith angle depends only on latitude. Local time of day fixed at 07:14:15 for radiative transfer computation ( $\sin(\text{time of day}) = 1/\pi$ ) 3: Zenith angle changing with latitude and time of day 4: Zenith angle and irradiance changing with season, latitude, and time of day (iforcing=inwp only)	
<b>albedo_type</b>	I	1		Type of surface albedo 1: based on soil type specific tabulated values (dry soil) 2: MODIS albedo	iforcing=inwp
<b>direct_albedo</b>	I	4		Direct beam surface albedo. Options mainly differ in terms of their solar zenith angle (SZA) dependency 1: SZA dependency following Ritter-Geleyn; applied to unconditionally all grid points 2: SZA dependency following Zaengl (pers. comm.). Same as 1 for water, but for 'rough surfaces' over land the direct albedo is not allowed to exceed the corresponding broadband diffuse albedo. 3: SZA dependency following Yang (2008) for snow-free land points. Same as 1 for water/ice and 2 for snow. 4: SZA dependency following Briegleb (1992) for snow-free land points. Same as 1 for water/ice and 2 for snow.	iforcing=inwp albedo_type=2



Parameter	Type	Default	Unit	Description	Scope
icld_overlap	I	2		Method for cloud overlap calculation in shortwave part of RRTM 1: maximum-random overlap 2: generalized overlap (Hogan, Illingworth, 2000) 3: maximum overlap 4: random overlap	iforcing=inwp inwp_radiation=1

Parameter	Type	Default	Unit	Description	Scope
irad_h2o	I	1		Switches for the concentration of radiative agents	
irad_co2		2		irad_xyz = 0: set to zero	
irad_ch4		3		irad_h2o = 1: vapor, cloud water and cloud ice from tracer variables	
irad_n2o		0		irad_co2 = 1: CO <sub>2</sub> from tracer variable	
irad_o3		2		irad_co2/ch4/n2o/o2/cfc11/cfc12 = 2: concentration given by	
irad_o2		2		vmr_co2/ch4/n2o/o2/cfc11/cfc12	
irad_cfc11		2		irad_ch4/n2o = 3: tanh-profile with surface concentration given by vmr_ch4/n2o	
irad_cfc12		2		irad_co2/cfc11/cfc12 = 4: time dependent concentration from greenhouse gas file	
				irad_ch4/n2o = 4: time dependent tanh-profile with surface concentration from greenhouse gas file	
				irad_o3 = 2: ozone climatology from MPI file	
				irad_o3 = 4: ozone clim for Aqua Planet Exp	
				irad_o3 = 6: ozone climatology with T5 geographical distribution and Fourier series for seasonal cycle	
				for run_nml/forcing = 3 (NWP)	
				irad_o3 = 7: GEMS ozone climatology (from IFS) for run_nml/forcing = 3 (NWP)	
				irad_o3 = 8: ozone climatology for AMIP	
				irad_o3 = 9: MACC ozone climatology (from IFS) for run_nml/forcing = 3 (NWP)	
				irad_o3 = 10: Linearized ozone chemistry (ART extension necessary) for run_nml/forcing = 3 (NWP)	

Parameter	Type	Default	Unit	Description	Scope
vmr_co2	R	348.0e-6		Volume mixing ratio of the radiative agents	
vmr_ch4		1650.0e-9			
vmr_n2o		306.0e-9			
vmr_o2		0.20946			
vmr_cfc11		214.5e-12			
vmr_cfc12		371.1e-12			
fh2o	R	1.		Scaling factors for concentrations used in radiation	run_nml/forcing=2 (ECHAM)
fco2		1.			
fch4		1.			
fn2o		1.			
fo3		1.			
fo2		1.			
fcfc		1.			
irad_aero	I	2		<p>Aerosols</p> <p>1: prognostic variable</p> <p>2: global constant</p> <p>3: externally specified</p> <p>5: Tanre aerosol climatology for <b>run_nml/forcing = 3 (NWP)</b></p> <p>6: Tegen aerosol climatology for <b>run_nml/forcing = 3 (NWP)</b> <b>AND. itopo = 1</b></p> <p>9: ART online aerosol radiation interaction, uses Tegen for aerosols not chosen to be represented in ART for <b>run_nml/forcing = 3 (NWP)</b> <b>AND. itopo = 1</b> <b>AND. iart=TRUE</b> <b>AND. iart_ari=1</b></p> <p>writes actual aerosol optical properties to output</p>	
irad_aero_diag	L	.FALSE.			

Parameter	Type	Default	Unit	Description	Scope
ighg	I	0		Select dynamic greenhouse gases scenario (read from file) 0 : select default gas volume mixing ratios - 1990 values (CMP5) 1 : transient CMP5 scenario from file	run_nml/forcing=2 (ECHAM)

Defined and used in: `src/namelist/mo_radiation_nml.f90`

#### 4.4.29 run\_nml

Parameter	Type	Default	Unit	Description	Scope
nsteps	I	-999		Number of time steps of this run. Allowed range is $\geq 0$ ; setting a value of 0 allows writing initial output (including internal remapping) without calculating time steps.	
dtime	R	600.0	s	time step. For real case runs the maximum allowable time step can be estimated as $1.8 \cdot \text{ndyn\_substeps} \cdot \overline{\Delta x} \text{ s km}^{-1}$ , where $\overline{\Delta x}$ is the average resolution in km and ndyn_substeps is the number of dynamics substeps set in nonhydrostatic_nml.	
ltestcase	L	.TRUE.		ndyn_substeps should not be increased beyond the default value 5.	
ldynamics	L	.TRUE.		Idealized testcase runs Compute adiabatic dynamic tendencies	

Parameter	Type	Default	Unit	Description	Scope
<b>iforcing</b>	I	0		Forcing of dynamics and transport by parameterized processes. Use positive indices for the atmosphere and negative indices for the ocean. 0: no forcing 1: Held-Suarez forcing 2: ECHAM forcing 3: NWP forcing 4: local diabatic forcing without physics 5: local diabatic forcing with physics -1: MPIOM forcing (to be done) Compute large-scale tracer transport Number of advected tracers handled by the large-scale transport scheme If set to .true. vertical nesting is switched on (i.e. variable number of vertical levels) Number of full levels (atm.) for each domain	lvert_nest=.TRUE. lvert_nest=.TRUE.
<b>ltransportracer</b>	L I	.FALSE. 0			
<b>lvert_nest</b>	L	.FALSE.			
<b>num_lev</b>	I(max_dom)	31			
nshift	I(max_dom)	0			
ltimer	L	.TRUE.		vertical half level of parent domain which coincides with upper boundary of the current domain <b>required for vertical refinement, which is not yet implemented</b> TRUE: Timer for monitoring the runtime of specific routines is on (FALSE = off)	
timers_level	I	1		TRUE: Timer for monitoring runtime of communication routines (FALSE = off)	
activate_sync_timers	L	F		controls how much printout is written during runtime.	
<b>msg_level</b>	I	10		For values less than 5, only the time step is written.	
msg_timestamp	L	.FALSE.		If .TRUE., precede output messages by time stamp.	

Parameter	Type	Default	Unit	Description	Scope
test_mode	I	0		Setting a value larger than 0 activates a dummy mode in which time stepping is changed into just doing iterations, and MPI communication is replaced by copying some value from the send buffer into the receive buffer (does not work with nesting and reduced radiation grid because the send buffer may then be empty on some PEs) Setting a value larger than 0 activates debug checks. Main switch for enabling/disabling components of the model output. One or more choices can be set (as an array of string constants). Possible choices are: <ul style="list-style-type: none"> <li>• “none”: switch off all output;</li> <li>• “nml”: new output mode (cf. <code>output_nml</code>);</li> <li>• “totint”: computation of total integrals.</li> <li>• “maxwinds”: write max. winds to separate ASCII file “maxwinds.log”.</li> </ul> If the output namelist parameter is not set explicitly, the default setting “nml” “totint” is assumed.	iequations = 3
debug_check_level	I	0			
output	C(:)	“nml”, ”totint”			
restart_filename	C			File name for restart/checkpoint files (containing keyword substitution patterns <code>&lt;gridfile&gt;</code> , <code>&lt;idom&gt;</code> , <code>&lt;rstarttime&gt;</code> , <code>&lt;mtype&gt;</code> ). default: ” <code>&lt;gridfile&gt;_restart_&lt;mtype&gt;_&lt;rstarttime&gt;.nc</code> ”.	

Parameter	Type	Default	Unit	Description	Scope
profiling_output	I	1		controls how profiling printout is written: TIMER_MODE_AGGREGATED=1, TIMER_MODE_DETAILED=2, TIMER_MODE_WRITE_FILES=3.	
lart	L	.FALSE.		Main switch which enables the treatment of atmospheric aerosol and trace gases (The ART package of KIT is needed for this purpose)	
check_uuid_gracefully	L	.FALSE.		If this flag is set to <code>.TRUE.</code> we give only warnings for non-matching UUIDs.	

Defined and used in: `src/namelist/mo_run_nml.f90`

#### 4.4.30 sleve\_nml (relevant if nonhydrostatic\_nml:ivctype=2)

Parameter	Type	Default	Unit	Description	Scope
<b>min_lay_thckn</b>	R	50	m	Layer thickness of lowermost layer; specifying zero or a negative value leads to constant layer thicknesses determined by top_height and nlev	
max_lay_thckn	R	25000	m	Maximum layer thickness below the height given by htop_thcknlimit (NWP recommendation: 400 m) <i>Use with caution! Too ambitious settings may result in numerically unstable layer configurations.</i>	
htop_thcknlimit	R	15000	m	Height below which the layer thickness does not exceed max_lay_thckn	
<b>top_height</b>	R	23500.0	m	Height of model top	
stretch_fac	R	1.0		Stretching factor to vary distribution of model levels; values <1 increase the layer thickness near the model top	
decay_scale_1	R	4000	m	Decay scale of large-scale topography component	

Parameter	Type	Default	Unit	Description	Scope
decay_scale_2	R	2500	m	Decay scale of small-scale topography component	
decay_exp	R	1.2		Exponent of decay function	
flat_height	R	16000	m	Height above which the coordinate surfaces are flat	
lread_smt	L	.FALSE.		read smoothed topography from file (TRUE) or compute internally (FALSE)	

Defined and used in: `src/namelist/mo_sleve_nml.f90`

4.4.31 **syosat\_nml**<sup>1</sup>

This namelist enables the RTTOV library incorporated into ICON for simulating satellite radiance and brightness temperatures. RTTOV is a radiative transfer model for nadir-viewing passive visible, infrared and microwave satellite radiometers, spectrometers and interferometers, see

<https://nwpsaf.eu/deliverables/rtm>

for detailed information.

Parameter	Type	Default	Unit	Description	Scope
lsynsat	L (max_dom)	.FALSE.		Main switch: Enables/disables computation of synthetic satellite imagery for each model domain.	
nlev_rttov	I	51		Number of RTTOV levels.	

Enabling the synsat module makes the following 32 two-dimensional output fields available:

`SYNMSG_RAD_CL_IR3.9`      `SYNMSG_BT_CL_IR3.9`      `SYNMSG_RAD_CL_WV6.2`      `SYNMSG_BT_CL_WV6.2`

<sup>1</sup>Important note: This feature is currently active for configuration `dwd+cray` only.



SYNMSG_RAD_CL_WV7.3	SYNMSG_BT_CL_WV7.3	SYNMSG_RAD_CL_IR8.7	SYNMSG_BT_CL_IR8.7
SYNMSG_RAD_CL_IR9.7	SYNMSG_BT_CL_IR9.7	SYNMSG_RAD_CL_IR10.8	SYNMSG_BT_CL_IR10.8
SYNMSG_RAD_CL_IR12.1	SYNMSG_BT_CL_IR12.1	SYNMSG_RAD_CL_IR13.4	SYNMSG_BT_CL_IR13.4
SYNMSG_RAD_CS_IR3.9	SYNMSG_BT_CS_IR3.9	SYNMSG_RAD_CS_WV6.2	SYNMSG_BT_CS_WV6.2
SYNMSG_RAD_CS_WV7.3	SYNMSG_BT_CS_WV7.3	SYNMSG_RAD_CS_IR8.7	SYNMSG_BT_CS_IR8.7
SYNMSG_RAD_CS_IR9.7	SYNMSG_BT_CS_IR9.7	SYNMSG_RAD_CS_IR10.8	SYNMSG_BT_CS_IR10.8
SYNMSG_RAD_CS_IR12.1	SYNMSG_BT_CS_IR12.1	SYNMSG_RAD_CS_IR13.4	SYNMSG_BT_CS_IR13.4

Here, RAD denotes radiance, BT brightness temperature, CL cloudy, and CS clear sky, supplemented by the channel name.

Defined and used in: `src/namelist/mo_synsat_nml.f90`

#### 4.4.32 time\_nml

Parameter	Type	Default	Unit	Description	Scope
calendar	I	1		Calendar type: 0=Julian/Gregorian 1=proleptic Gregorian 2=30day/month, 360day/year	

Parameter	Type	Default	Unit	Description	Scope
dt_restart	R	86400.*30.	s	Length of restart cycle in seconds. This namelist parameter specifies how long the model runs until it saves its state to a file and stops. Later, the model run can be resumed, s. t. a simulation over a long period of time can be split into a chain of restarted model runs. Note that the frequency of writing restart files is controlled by <code>io_nml:dt_checkpoint</code> . Only if the value of <code>dt_checkpoint</code> resulting from model default or user's specification is longer than <code>dt_restart</code> , it will be reset (by the model) to <code>dt_restart</code> so that at least one restart file is generated during the restart cycle. If <code>dt_restart</code> is larger than but not a multiple of <code>dt_checkpoint</code> , restart file will <i>not</i> be generated at the end of the restart cycle. Initial date and time of the simulation	
ini_datetime_string	C	'2008-09-01T00:00:00Z'		End date and time of the simulation	
end_datetime_string	C	'2008-09-01T01:40:00Z'		.TRUE., if time loop shall start with step 0 regardless whether we are in a standard run or in a restarted run (which means re-initialized run).	
is_relative_time	L	.FALSE.			

**Length of the run** If "nsteps" in run\_nml is positive, then `nsteps*dtime` is used to compute the end date and time of the run. Else the initial date and time, the end date and time, `dt_restart`, as well as the time step are used to compute "nsteps".

## 4.4.33 transport\_nml (used if run\_nml/ltransport=.TRUE.)

Parameter	Type	Default	Unit	Description	Scope
<b>lvadv_tracer</b>	L	.TRUE.		TRUE : compute vertical tracer advection FALSE: do not compute vertical tracer advection	
<b>ihadv_tracer</b>	I(ntracer)	2		Tracer specific method to compute horizontal advection: 0: no horiz. transport (note that the specific tracer quantity $q$ is kept constant and not tracer mass $\rho q$ ) 1: upwind (1st order) 2: Miura (2nd order, linear reconstr.) 3: Miura3 (quadr. or cubic reconstr.) 4: FFSL (quadr. or cubic reconstr.) 5: hybrid Miura3/FFSL (quadr. or cubic reconstr.) 20: miura (2nd order, lin. reconstr.) with subcycling 22: combination of miura and miura with subcycling 32: combination of miura3 and miura with subcycling 42: combination of FFSL and miura with subcycling 52: combination of hybrid FFSL/Miura3 with subcycling Subcycling means that the integration from time step $n$ to $n+1$ is splitted into substeps to meet the stability requirements. For NWP runs, substepping is generally applied above $z = 22$ km (see nonhydrostatic_nml/hbot_qvsubstep).	lsq_high_ord $\in [2,3]$ lsq_high_ord $\in [2,3]$ lsq_high_ord $\in [2,3]$

Parameter	Type	Default	Unit	Description	Scope
lvadv_tracer	I(ntracer)	3		Tracer specific method to compute vertical advection: 0: no vert. transport (note that tracer mass $\rho q$ instead of the specific tracer quantity $q$ is kept constant. This differs from the behaviour in horizontal direction!) 1: upwind (1st order) 3: ppm_cfl (3 <sup>rd</sup> order, handles CFL > 1) 30: ppm (3 <sup>rd</sup> order, CFL ≤ 1) Type of TKE advection 0: no TKE advection 1: vertical advection only 2: vertical and horizontal advection Time splitting method TRUE: second order Strang splitting FALSE: first order Godunov splitting	lvadv_tracer=TRUE
lvadv_tke	I	0		Two purposes: - used for selecting those tracers which should be initialized for idealized testcases. - used for tracer output names. In some idealized cases tracers are named ' $Qx$ ', with $x$ being a 1-digit integer taken from <code>ctracer_list</code> .	inwp_turb=1
lvstrang	L	.FALSE.			
ctracer_list	C	"			[nh/ha]_test_name= 'jabw','PA','DF' iforcing≠ inwp, iechem
npassive_tracer	I	0		number of additional passive tracers which have no sources and are transparent to any physical process (no effect). Passive tracers are named Qpassive.ID, where ID is a number between ntracer and ntracer+npassive_tracer. <b>NOTE:</b> By default, limiters are switched of for passive tracers and the scheme 52 is selected for horizontal advection.	

Parameter	Type	Default	Unit	Description	Scope
init_formula	C	, ,		Comma-separated list of initialization formulas for additional passive tracers.	npassive_tracer $\neq$ 0
<b>itype_hlimit</b>	I(ntracer)	4		Type of limiter for horizontal transport: 0: no limiter 3: monotonous flux limiter 4: positive definite flux limiter	
<b>itype_vlimit</b>	I(ntracer)	1		Type of limiter for vertical transport: 0: no limiter 1: semi-monotone slope limiter 2: monotonous slope limiter 4: positive definite flux limiter	
niter_fct	I	1		number of iterations of monotone flux correction procedure (experimental!)	itype_hlimit = 3
beta_fct	R	1.005		factor of allowed over-/undershooting in monotonous limiter	itype_hlimit = 3
iord_backtraj	I	1		order of backward trajectory calculation: 1: first order 2: second order (iterative; currently 1 iteration hardcoded; experimental!)	ihadv_tracer='miura'
igrad_c_miura	I	1		Method for gradient reconstruction at cell center for 2nd order miura scheme 1: Least-squares (linear, non-consv) 2: Green-Gauss	ihadv_tracer=2 ivadv_tracer=3
ivcfl_max	I	5		determines stability range of vertical PPM-scheme in terms of the maximum allowable CFL-number	
llsq_svd	L	.TRUE.		use QR decomposition (FALSE) or SV decomposition (TRUE) for least squares design matrix A	
lclip_tracer	L	.FALSE.		Clipping of negative values	

Defined and used in: `src/namelist/mo_advection_nml.f90`

**4.4.34 turbdiff\_nml**

Parameter	Type	Default	Unit	Description	Scope
imode_turb	I	1		Mode of solving the TKE equation for atmosph. layers: 0: diagnostic equation 1: prognostic equation (current version) 2: prognostic equation (intrinsically positive definite) Same as <i>imode_turb</i> but only for the transfer layer	
imode_tran	I	0		Mode of water cloud representation in turbulence for atmosph. layers: -1: ignoring cloud water completely (pure dry scheme) 0: no clouds considered (all cloud water is evaporated)	
icldm_turb	I	2		1: only grid scale condensation possible 2: also sub grid (turbulent) condensation considered Same as <i>icldm_turb</i> but only for the transfer layer	
icldm_tran	I	2		type of water cloud diagnosis within the turbulence scheme: 1: employing a scheme based on relative humidity 2: employing a statistical saturation adjustment	icldm_turb=2 or icldm_tran=2
itype_wcld	I	2		Type of shear forcing used in turbulence: 0: only vertical shear of horizontal wind 1: previous plus horizontal shear correction 2: previous plus shear from vertical velocity 3: same as option 1, but (when combined with ltkeshs=.TRUE.) scaling of coarse-grid horizontal shear production term with $\frac{1}{\sqrt{Ri}}$	
itype_sher	I	0			

Parameter	Type	Default	Unit	Description	Scope
ltkeshs	L	.FALSE.		Include correction term for coarse grids in horizontal shear production term (needed at non-convective-resolving model resolutions in order to get a non-negligible impact) Consider TKE-production by sub grid SSO wakes	itype_sher $\geq 1$
ltkesso	L	.FALSE.		Consider TKE-production by sub grid convective plumes (inactive)	inwp_sso = 1
ltkecon	L	.FALSE.		Consider TKE-production by separated horizontal shear eddies (inactive)	inwp_conv = 1
ltmpcor	L	.FALSE.		Consider thermal TKE sources in enthalpy equation	
lsflend	L	.TRUE.		Use lower flux condition for vertical diffusion calculation (TRUE) instead of a lower concentration condition (FALSE)	
lexpcor	L	.FALSE.		Explicit corrections of implicitly calculated vertical diffusion of non-conservative scalars that are involved in sub grid condensation processes	
tur_len	R	500.0	m	Asymptotic maximal turbulent distance ( $\kappa * tur\_len$ is the integral turbulent master length scale)	
pat_len	R	100.0	m	Effective length scale of thermal surface patterns controlling TKE-production by sub grid kata/ana-batic circulations. In case of $pat\_len = 0$ , this production is switched off.	
c.diff	R	0.2	1	Length scale factor for vertical diffusion of TKE. In case of $c\_diff = 0$ , TKE is not diffused vertically.	
a.stab	R	0.0	1	Factor for stability correction of turbulent length scale. In case of $a\_stab = 0$ , the turbulent length scale is not reduced for stable stratification.	

Parameter	Type	Default	Unit	Description	Scope
a_hshr	R	0.20	1	Length scale factor for the separated horizontal shear mode. In case of $a\_hshr = 0$ , this shear mode has no effect.	ltkeshs=.TRUE.
alpha0	R	0.0123	1	Lower bound of velocity-dependent Charnock parameter	
alpha0_max	R	0.0335	1	Upper bound of velocity-dependent Charnock parameter. Setting this parameter to 0.0335 or higher values implies unconstrained velocity dependence	
tkhmin	R	0.75	m <sup>2</sup> /s	Scaling factor for minimum vertical diffusion coefficient (proportional to $1/\sqrt{Ri}$ ) for heat and moisture	
tkmmin	R	0.75	m <sup>2</sup> /s	Scaling factor for minimum vertical diffusion coefficient (proportional to $1/\sqrt{Ri}$ ) for momentum	
tkmmin_strat	R	5	m <sup>2</sup> /s	Enhanced scaling factor for minimum vertical diffusion coefficient (proportional to $1/\sqrt{Ri}$ ) for momentum, valid in the stratosphere above 30 km	
tkhmin_strat	R	5	m <sup>2</sup> /s	Enhanced scaling factor for minimum vertical diffusion coefficient (proportional to $1/\sqrt{Ri}$ ) for heat and moisture, valid in the stratosphere above 30 km	
itype_synd	I	2		Type of diagnostics of synoptic near surface variables: 1: Considering the mean surface roughness of a grid box 2: Considering a fictive surface roughness of a SYNOP lawn	
rlam_heat	R	1.0	1	Scaling factor of the laminar boundary layer for heat (scalars). The larger rlam_heat, the larger is the laminar resistance.	



Parameter	Type	Default	Unit	Description	Scope
rat_sea	R	10.0	1	Ratio of laminar scaling factors for scalars over sea and land. The larger rat_sea, the larger is the laminar resistance for a sea surface compared to a land surface.	
tkesmot	R	0.15	1	Time smoothing factor within [0, 1] for TKE.	
frcsmot	R	0.0	1	In case of <i>tkesmot</i> = 0, no smoothing is active. Vertical smoothing factor within [0, 1] for TKE forcing terms. In case of <i>frcmot</i> = 0, no smoothing is active.	
imode_frcsmot	I	1		1 = apply vertical smoothing (if <i>frcsmot</i> > 0) uniformly over the globe 2 = restrict vertical smoothing to the tropics (reduces the moist bias in the tropics while avoiding adverse effects on NWP skill scores in the extratropics)	
impls	R	1.20	1	Implicit weight near the surface (maximal value)	
implt	R	0.75	1	Implicit weight near top of the atmosphere (minimal value)	
lconst_z0	L	.FALSE.		TRUE: horizontally homogeneous roughness length z0	
const_z0	R	0.001	m	value for horizontally homogeneous roughness length z0	lconst_z0=.TRUE.
ldiff_qi	L	.FALSE.		Turbulent diffusion of cloud ice, if .TRUE.	
itype_tran	I	2		type of surface-atmosphere transfer using the profile values of the lowest main level instead of the mean value of the lowest layer for surface flux calculations	
lprfcor	L	.FALSE.		nonlocal calculation of vertical gradients used for turbul. diff.	
lnonloc	L	.FALSE.			

Parameter	Type	Default	Unit	Description	Scope
lfreeslip	L	.FALSE.		.TRUE.: use a free-slip lower boundary condition, i.e. neither momentum nor heat/moisture fluxes (use for idealized runs only!)	
lcpfluc	L	.FALSE.		consideration of fluctuations of the heat capacity of air	

Defined and used in: `src/namelist/mo_turbdiff_nml.f90`

#### 4.4.35 vdiff\_nml

Parameter	Type	Default	Unit	Description	Scope
lsfc_mon_flux	L	.TRUE.		Switch on surface momentum flux.	lvdiff = .TRUE.
lsfc_heat_flux	L	.TRUE.		Switch on surface sensible and latent heat flux.	lvdiff = .TRUE.

Defined and used in: `src/namelist/mo_vdiff_nml.f90`

## 4.5 Ocean-specific namelist parameters

### 4.5.1 ocean\_physics\_nml

Parameter	Type	Default	Unit	Description	Scope
i_sea_ice	I	1		0: No sea ice, 1: Include sea ice	
richardson_factor_tracer	I	0.5e-5	m/s	.FALSE.: compute drag only	
richardson_factor_veloc	I	0.5e-5	m/s		
l_constant_mixing	L	.FALSE.			

#### 4.5.2 sea\_ice\_nml (relevant if run\_nml/forcing=2 (ECHAM))

Parameter	Type	Default	Unit	Description	Scope
i_ice_therm	I	2		Switch for thermodynamic model: 1: Zero-layer model 2: Two layer Winton (2000) model 3: Zero-layer model with analytical forcing (for diagnostics) 4: Zero-layer model for atmosphere-only runs (for diagnostics)	In an ocean run i_sea_ice must be $i_j=1$ . In an atmospheric run the ice surface type must be defined.
i_ice_dyn	I	0		Switch for sea-ice dynamics: 0: No dynamics 1: FEM dynamics (from AWI)	
i_ice_albedo	I	1		Switch for albedo model. Only one is implemented so far.	
i_Qio_type	I	2		Switch for ice-ocean heat-flux calculation method: 1: Proportional to ocean cell thickness (like MPI-OM) 2: Proportional to speed difference between ice and ocean	Defaults to 1 when i_ice_dyn=0 and 2 otherwise.
kice	I	1		Number of ice classes (must be one for now)	
hnull	R	0.5	m	Hibler's $h_0$ parameter for new-ice growth.	
hmin	R	0.05	m	Minimum sea-ice thickness allowed.	
ramp_wind	R	10	days	Number of days it takes the wind to reach correct strength. Only used at the start of an OMIP/NCEP simulation (not after restart).	

#### 4.6 Namelist parameters for testcases (NAMELIST\_ICON)

The ICON model code includes several experiments, so-called test cases, for the shallow water model as well as the 3-dimensional atmosphere. Depending on the specified experiment, initial conditions and boundary conditions are computed internally.

#### 4.6.1 ha\_testcase\_nml (Scope: ltestcase=.TRUE. and iequations=[0,1,2] in run\_nml)

Parameter	Type	Default	Unit	Description	Scope
cctest_name	C	'JWw'		<p>Name of test case:</p> <p>'SW_GW': gravity wave</p> <p>'USBR': unsteady solid body rotation</p> <p>'Will_2': Williamson test 2</p> <p>'Will_3': Williamson test 3</p> <p>'Will_5': Williamson test 5</p> <p>'Will_6': Williamson test 6</p> <p>'GW': gravity wave (nlev=20 only!)</p> <p>'LDF': local diabatic forcing test without physics</p> <p>'LDF-Moist': local diabatic forcing test with physics initialised with zonal wind field</p> <p>'HS': Held-Suarez test</p> <p>'JWs': Jablonowski-Will. steady state</p> <p>'JWw': Jablonowski-Will. wave test</p> <p>'JWw-Moist': Jablonowski-Will. wave test including moisture</p> <p>'APE': aqua planet experiment</p> <p>'MRW': mountain induced Rossby wave</p> <p>'MRW2': modified mountain induced Rossby wave</p> <p>'PA': pure advection</p> <p>'SV': stationary vortex</p> <p>'DF1': deformational flow test 1</p> <p>'DF2': deformational flow test 2</p> <p>'DF3': deformational flow test 3</p> <p>'DF4': deformational flow test 4</p> <p>'RH': Rossby-Haurwitz wave test</p>	<p>lshallow_water=.TRUE.</p> <p>lshallow_water=.TRUE.</p> <p>lshallow_water=.TRUE.</p> <p>lshallow_water=.TRUE.</p> <p>lshallow_water=.TRUE.</p> <p>lshallow_water=.TRUE.</p> <p>lshallow_water=.FALSE.</p> <p>lshallow_water=.FALSE.</p> <p>and iforcing=4</p> <p>lshallow_water=.FALSE., and iforcing=5</p> <p>lshallow_water=.FALSE.</p> <p>lshallow_water=.FALSE.</p> <p>lshallow_water=.FALSE.</p> <p>lshallow_water=.FALSE.</p> <p>lshallow_water=.FALSE.</p> <p>lshallow_water=.FALSE.</p> <p>lshallow_water=.FALSE.</p> <p>lshallow_water=.FALSE.</p> <p>lshallow_water=.FALSE., ntracer = 2</p> <p>lshallow_water=.FALSE.</p>

Parameter	Type	Default	Unit	Description	Scope
rotate_axis_deg	R	0.0	deg	Earth's rotation axis pitch angle	ctest_name= 'Will_2', 'Will_3', 'JWs', 'JWw', 'PA', 'DF1234'
gw_brunt_vais	R	0.01	1/s	Brunt Vaisala frequency	ctest_name= 'GW'
gw_u0	R	0.0	m/s	zonal wind parameter	ctest_name= 'GW'
gw_lon_deg	R	180.0	deg	longitude of initial perturbation	ctest_name= 'GW'
gw_lat_deg	R	0.0	deg	latitude of initial perturbation	ctest_name= 'GW'
jw_uptb	R	1.0	m/s	amplitude of the wave perturbation	ctest_name= 'JWw'
			(?)		
mountctr_lon_deg	R	90.0	deg	longitude of mountain peak	ctest_name= 'MRW(2)'
mountctr_lat_deg	R	30.0	deg	latitude of mountain peak	ctest_name= 'MRW(2)'
mountctr_height	R	2000.0	m	mountain height	ctest_name= 'MRW(2)'
mountctr_half_width	R	1500000.0	m	mountain half width	ctest_name= 'MRW(2)'
mount_u0	R	20.0	m/s	wind speed for MRW cases	ctest_name= 'MRW(2)'
rh_wavenum	I	4		wave number	ctest_name= 'RH'
rh_init_shift_deg	R	0.0	deg	pattern shift	ctest_name= 'RH'
rhs_init_type	I	1		Choice of initial condition for the Held-Suarez test. 1: the zonal state defined in the JW's test case; other integers: isothermal state (T=300 K, ps=1000 hPa, u=v=0.)	ctest_name= 'HS'
lhs_vn_ptb	L	.TRUE.		Add random noise to the initial wind field in the Held-Suarez test.	ctest_name= 'HS'
hs_vn_ptb_scale	R	1.	m/s	Magnitude of the random noise added to the initial wind field in the Held-Suarez test.	ctest_name= 'HS'
lrh_linear_pres	L	.FALSE.		Initialize the relative humidity using a linear function of pressure.	ctest_name= 'JWw-Moist', 'APE', 'LDF-Moist'
rh_at_1000hpa	R	0.75		relative humidity	ctest_name= 'JWw-Moist', 'APE', 'LDF-Moist'
				0, 1	
limit_tracer_fv	L	.TRUE.		at 1000 hPa Finite volume initialization for tracer fields	ctest_name= 'PA'

Parameter	Type	Default	Unit	Description	Scope
ape_sst_case	C	'sst1'		SST distribution selection 'sst1': Control experiment 'sst2': Peaked experiment 'sst3': Flat experiment 'sst4': Control-5N experiment 'sst_qobs': Qobs SST distribution exp 'sst_ice': Control SST distribution with -1.8 C above 64 N/S.	cstest_name='APE'
ildf_init_type	I	0		Choice of initial condition for the Local diabatic forcing test. 1: the zonal state defined in the JWs test case; other: isothermal state (T=300 K, ps=1000 hPa, u=v=0.)	cstest_name= 'LDF'
ldf_symm	L	.TRUE.		Shape of local diabatic forcing: .TRUE.: local diabatic forcing symmetric about the equator (at 0 N) .FALSE.: local diabatic forcing asym. about the equator (at 30 N)	cstest_name= 'LDF', 'LDF-Moist'

Defined and used in: src/testcases/mo\_ha\_testcases.f90

#### 4.6.2 nh\_testcase\_nml (Scope: ltestcase=.TRUE. and iequations=3 in run\_nml)

Parameter	Type	Default	Unit	Description	Scope
nh_test_name	C	'jabw'		testcase selection 'zero': no orography 'bell': bell shaped mountain at 0E,0N 'schaer': hilly mountain at 0E,0N 'jabw': Initializes the full Jablonowski Williamson test case. 'jabw_s': Initializes the Jablonowski Williamson steady state test case.	

Parameter	Type	Default	Unit	Description	Scope
				<p><b>'jabw_m'</b>: Initializes the Jablonowski Williamson test case with a mountain instead of the wind perturbation (specify <code>mount_height</code>).</p> <p><b>'mrw_nh'</b>: Initializes the full Mountain-induced Rossby wave test case.</p> <p><b>'mrw2_nh'</b>: Initializes the modified mountain-induced Rossby wave test case.</p> <p><b>'mwbr_const'</b>: Initializes the mountain wave with two layers test case. The lower layer is isothermal and the upper layer has constant brunt vaisala frequency. The interface has constant pressure.</p> <p><b>'PA'</b>: Initializes the pure advection test case.</p> <p><b>'HS_nh'</b>: Initializes the Held-Suarez test case. At the moment with an isothermal atmosphere at rest (<math>T=300K</math>, <math>ps=1000hPa</math>, <math>u=v=0</math>, <math>topography=0.0</math>).</p> <p><b>'HS_jw'</b>: Initializes the Held-Suarez test case with Jablonowski Williamson initial conditions and zero topography.</p> <p><b>'APE_nwp, APE_echam, APE_nh, APEc_nh, '</b>: Initializes the APE experiments. With the <code>jabw</code> test case, including moisture.</p> <p><b>'wk82'</b>: Initializes the Weisman Klemp test case</p> <p><b>'g_lim_area'</b>: Initializes a series of general limited area test cases: <code>itype_atmos_ana</code> determines the atmospheric profile, <code>itype_anaprov_uv</code> determines the wind profile and <code>itype_topo_ana</code> determines the topography</p> <p><b>'dcmip_pa_12'</b>: Initializes Hadley-like meridional circulation pure advection test case.</p>	<p><code>limited_area = .TRUE.</code></p>

Parameter	Type	Default	Unit	Description	Scope
jw_up	R	1.0	m/s	' <b>dcmip_rest_200</b> ': atmosphere at rest test (Schaer-type mountain)	lcoriolis = .FALSE.
				' <b>dcmip_mw_2x</b> ': nonhydrostatic mountain waves triggered by Schaer-type mountain	lcoriolis = .FALSE.
u0_mrw	R	20.0	m/s	' <b>dcmip_gw_31</b> ': nonhydrostatic gravity waves triggered by a localized perturbation (nonlinear)	l_limited_area = .TRUE. and lcoriolis = .FALSE. lcoriolis = .TRUE.
				' <b>dcmip_gw_32</b> ': nonhydrostatic gravity waves triggered by a localized perturbation (linear)	
mount_height_mrw	R	2000.0	m	' <b>dcmip_tc_51</b> ': tropical cyclone test case with 'simple physics' parameterizations ( <b>not yet implemented</b> )	lcoriolis = .TRUE.
				' <b>dcmip_tc_52</b> ': tropical cyclone test case with full physics in Aqua-planet mode	is_plane_torus = .TRUE.
mount_half_width	R	1500000.0	m	' <b>CBL</b> ': convective boundary layer simulations for LES package on torus (doubly periodic) grid	nh_test_name='jabw'
				amplitude of the u-perturbation in jabw test case	nh_test_name='mrw(2)_nh' and 'mwbr_const'
mount_lonctr_mrw_deg	R	90.	deg	wind speed for mrw(2) and mwbr_const cases	nh_test_name='mrw(2)_nh' and 'mwbr_const'
				maximum mount height in mrw(2) and mwbr_const	nh_test_name='mrw(2)_nh' and 'mwbr_const'
mount_latctr_mrw_deg	R	30.	deg	half width of mountain in mrw(2), mwbr_const and bell	nh_test_name='mrw(2)_nh' and 'bell'
				lon of mountain center in mrw(2) and mwbr_const	nh_test_name='mrw(2)_nh' and 'mwbr_const'
				lat of mountain center in mrw(2) and mwbr_const	nh_test_name='mrw(2)_nh' and 'mwbr_const'



Parameter	Type	Default	Unit	Description	Scope
temp_i_mwbr_const	R	288.0	K	temp at isothermal lower layer for mwbr_const case	nh_test_name= 'mwbr_const'
p_int_mwbr_const	R	70000.	Pa	pres at the interface of the two layers for mwbr_const case	nh_test_name= 'mwbr_const'
bruntvais_u_mwbr_const	R	0.025	1/s	constant brunt vaissala frequency at upper layer for mwbr_const case	nh_test_name= 'mwbr_const'
mount_height	R	100.0	m	peak height of mountain	nh_test_name= 'bell'
layer_thickness	R	-999.0	m	thickness of vertical layers	If layer_thickness < 0, the vertical level distribution is read in from externally given HYB_PARAMS_XX.
n_flat_level	I	2		level number for which the layer is still flat and not terrain-following	layer_thickness > 0
nh_u0	R	0.0	m/s	initial constant zonal wind speed	nh_test_name = 'bell'
nh_t0	R	300.0	K	initial temperature at lowest level	nh_test_name = 'bell'
nh_brunt_vais	R	0.01	1/s	initial Brunt-Vaisala frequency	nh_test_name = 'bell'
torus_domain_length	R	100000.0	m	length of slice domain	nh_test_name = 'bell', lplane=TRUE.
rotate_axis_deg	R	0.0	deg	Earth's rotation axis pitch angle	nh_test_name= 'PA'
lhs_nh_vn_ptb	L	.TRUE.		Add random noise to the initial wind field in the Held-Suarez test.	nh_test_name= 'HS_nh'
lhs_fric_heat	L	.FALSE.		add frictional heating from Rayleigh friction in the Held-Suarez test.	nh_test_name= 'HS_nh'
hs_nh_vn_ptb_scale	R	1.	m/s	Magnitude of the random noise added to the initial wind field in the Held-Suarez test.	nh_test_name= 'HS_nh'
rh_at_1000hpa	R	0.7	1	relative humidity at 1000 hPa	nh_test_name= 'jabw', nh_test_name= 'mrw'
qv_max	R	20.e-3	kg/kg	specific humidity in the tropics	nh_test_name= 'jabw', nh_test_name= 'mrw'

Parameter	Type	Default	Unit	Description	Scope
ape_sst_case	C	'sst1'		SST distribution selection 'sst1': Control experiment 'sst2': Peaked experiment 'sst3': Flat experiment 'sst4': Control-5N experiment 'sst_qobs': Qobs SST distribution exp. 'sst_const': constant SST	nh_test_name='APE_nwp', 'APE_echam'
ape_sst_val	R	29.0	degC	aqua planet SST for ape_sst_case='sst_const'	nh_test_name= 'APE_nwp', 'APE_echam'
limit_tracer_fv	L	.TRUE.		Finite volume initialization for tracer fields	pure advection tests, only
lcoupled_rho	L	.FALSE.		Integrate density equation 'offline'	pure advection tests, only
qv_max_wk	R	0.014	Kg/kg	maximum specific humidity near the surface, range 0.012 - 0.016 used to vary the buoyancy	nh_test_name='wk82'
u_infty_wk	R	20.	m/s	zonal wind at infinity height range 0. - 45. used to vary the wind shear	nh_test_name='wk82'
bub_amp	R	2.	K	maximum amplitud of the thermal perturbation	nh_test_name='wk82'
bubctr_lat	R	0.	deg	latitude of the center of the thermal perturbation	nh_test_name='wk82'
bubctr_lon	R	90.	deg	longitude of the center of the thermal perturbation	nh_test_name='wk82'
bubctr_z	R	1400.	m	height of the center of the thermal perturbation	nh_test_name='wk82'
bub_hor_width	R	10000.	m	horizontal radius of the thermal perturbation	nh_test_name='wk82'
bub_ver_width	R	1400.	m	vertical radius of the thermal perturbation	nh_test_name='wk82'
itype_atmo_ana	I	1		kind of atmospheric profile: 1 piecewise N constant layers 2 piecewise polytropic layers	nh_test_name= 'g_lim_area'

Parameter	Type	Default	Unit	Description	Scope
itype_anaprof_uv	I	1		kind of wind profile: 1 piecewise linear wind layers 2 constant zonal wind 3 constant meridional wind	nh_test_name= 'g_lim_area'
itype_topo_ana	I	1		kind of orography: 1 schaefer test case mountain 2 gaussian_2d mountain 3 gaussian_3d mountain any other no orography	nh_test_name= 'g_lim_area'
nlayers_nconst	I	1		Number of the desired layers with a constant Brunt-Vaisala-frequency	nh_test_name= 'g_lim_area' and itype_atmo_ana=1
p_base_nconst	R	100000.	Pa	pressure at the base of the first N constant layer	nh_test_name= 'g_lim_area' and itype_atmo_ana=1
theta0_base_nconst	R	288.	K	potential temperature at the base of the first N constant layer	nh_test_name= 'g_lim_area' and itype_atmo_ana=1
h_nconst	R(nlayers_nconst)	0., 1500., 12000.	m	height of the base of each of the N constant layers	nh_test_name= 'g_lim_area' and itype_atmo_ana=1
N_nconst	R(nlayers_nconst)	0.01	1/s	Brunt-Vaisala-frequency at each of the N constant layers	nh_test_name= 'g_lim_area' and itype_atmo_ana=1
rh_nconst	R(nlayers_nconst)	0.5	%	relative humidity at the base of each N constant layers	nh_test_name= 'g_lim_area' and itype_atmo_ana=1
rhgr_nconst	R(nlayers_nconst)	0.	%	relative humidity gradient at each of the N constant layers	nh_test_name= 'g_lim_area' and itype_atmo_ana=1
nlayers_poly	I	2		Number of the desired layers with constant gradient temperature	nh_test_name= 'g_lim_area' and itype_atmo_ana=2

Parameter	Type	Default	Unit	Description	Scope
p_base_poly	R	100000.	Pa	pressure at the base of the first polytropic layer	nh_test_name= 'g_lim_area' and itype_atmo_ana=2
h_poly	R(nlayers -poly)	0., 12000.	m	height of the base of each of the polytropic layers	nh_test_name= 'g_lim_area' and itype_atmo_ana=2
t_poly	R(nlayers -poly)	288., 213.	K	temperature at the base of each of the polytropic layers	nh_test_name= 'g_lim_area' and itype_atmo_ana=2
rh_poly	R(nlayers -poly)	0.8, 0.2	%	relative humidity at the base of each of the polytropic layers	nh_test_name= 'g_lim_area' and itype_atmo_ana=2
rhgr_poly	R(nlayers -poly)	5.e-5, 0.	%	relative humidity gradient at each of the polytropic layers	nh_test_name= 'g_lim_area' and itype_atmo_ana=2
nlayers_linwind	I	2		Number of the desired layers with constant U gradient	nh_test_name= 'g_lim_area' and itype_anaprof_uv=1
h_linwind	R(nlayers _linwind)	0., 2500.	m	height of the base of each of the linear wind layers	nh_test_name= 'g_lim_area' and itype_anaprof_uv=1
u_linwind	R(nlayers _linwind)	5, 10.	m/s	zonal wind at the base of each of the linear wind layers	nh_test_name= 'g_lim_area' and itype_anaprof_uv=1
ugr_linwind	R(nlayers _linwind)	0., 0.	1/s	zonal wind gradient at each of the linear wind layers	nh_test_name= 'g_lim_area' and itype_anaprof_uv=1
velconst	R	20.	m/s	constant zonal/meridional wind (itype_anaprof_uv=2,3)	nh_test_name= 'g_lim_area' and itype_anaprof_uv=2,3
mount_lonc_deg	R	90.	deg	longitud of the center of the mountain	nh_test_name= 'g_lim_area'

Parameter	Type	Default	Unit	Description	Scope
mount_latc_deg	R	0.	deg	latitude of the center of the mountain	nh_test_name= 'g_lim_area'
schaer_h0	R	250.	m	h0 parameter for the schaefer mountain	nh_test_name= 'g_lim_area' and itype_topo_ana=1
schaer_a	R	5000.	m	-a- parameter for the schaefer mountain, also half width in the north and south side of the finite ridge to round the sharp edges	nh_test_name= 'g_lim_area' and itype_topo_ana=1,2
schaer_lambda	R	4000.	m	lambda parameter for the schaefer mountain	nh_test_name= 'g_lim_area' and itype_topo_ana=1
lshear_dcmip	L	FALSE		run dcmip_mw_2x with/without vertical wind shear FALSE: dcmip_mw_21: non-sheared TRUE : dcmip_mw_22: sheared	nh_test_name= 'g_lim_area' and itype_topo_ana=1
halfwidth_2d	R	10000.	m	half length of the finite ridge in the north-south direction	nh_test_name= 'g_lim_area' and itype_topo_ana=1,2
m_height	R	1000.	m	height of the mountain	nh_test_name= 'g_lim_area' and itype_topo_ana=2,3
m_width_x	R	5000.	m	half width of the gaussian mountain in the east-west direction	nh_test_name= 'g_lim_area' and itype_topo_ana=2,3
m_width_y	R	5000.	m	half width in the north-south direction in the rounding of the finite ridge (gaussian_2d) half width of the gaussian mountain in the north-south direction	nh_test_name= 'g_lim_area' and itype_topo_ana=2,3
gw_u0	R	0.	m/s	maximum amplitude of the zonal wind	nh_test_name= 'dcmip_gw_3X'
gw_clat	R	90.	deg	Lat of perturbation center	nh_test_name= 'dcmip_gw_3X'

Parameter	Type	Default	Unit	Description	Scope
gw_delta_temp	R	0.01	K	maximum temperature perturbation	nh_test_name= 'dcmip_gw_32'
u_cbl(2)	R	0:0	m/s and 1/s	to prescribe initial zonal velocity profile for convective boundary layer simulations where u_cbl(1) sets the constant and u_cbl(2) sets the vertical gradient	nh_test_name=CBL
v_cbl(2)	R	0:0	m/s and 1/s	to prescribe initial meridional velocity profile for convective boundary layer simulations where v_cbl(1) sets the constant and v_cbl(2) sets the vertical gradient	nh_test_name=CBL
th_cbl(2)	R	290:0.006	K and K/m	to prescribe initial potential temperature profile for convective boundary layer simulations where th_cbl(1) sets the constant and th_cbl(2) sets the gradient	nh_test_name=CBL

Defined and used in: src/testcases/mo\_nh\_testcases.f90

## 4.7 External data

### 4.7.1 extpar\_nml (Scope: itopo=1 in run\_nml)

Parameter	Type	Default	Unit	Description	Scope
<b>itopo</b>	I	0		0: analytical topography/ext. data 1: topography/ext. data read from file	
<b>n_iter_smooth_topo</b>	I(n_dom)	0		iterations of topography smoother	itopo = 1
fac_smooth_topo	R	0.015625		pre-factor of topography smoother	n_iter_smooth_topo > 0
hgt_diff_max_smooth_topo	R	0.	m	RMS height difference to neighbor grid points at which the smoothing pre-factor fac_smooth_topo reaches its maximum value (linear proportionality for weaker slopes)	n_iter_smooth_topo > 0

Parameter	Type	Default	Unit	Description	Scope
heightdiff_threshold	R(n_dom)	3000.	m	height difference between neighboring grid points above which additional local nabl2 diffusion is applied	
l_emiss	L	.TRUE.		read and use external surface emissivity map	
extpar_filename	C			Filename of external parameter input file, default: "<path>extpar.<gridfile>". May contain the keyword <path> which will be substituted by model_base_dir.	itopo = 1
extpar_varnames_map_file	C	,		Filename of external parameter dictionary, This is a text file with two columns separated by whitespace, where left column: NetCDF name, right column: GRIB2 short name. It is required, if external parameter are read from a file in GRIB2 format.	

Defined and used in: src/namelists/mo\_extpar\_nm1.f90

## 4.8 External packages

## 4.9 Information on vertical level distribution

If no vertical sleeve coordinate is chosen (ivctype / =2), the hydrostatic and nonhydrostatic models need hybrid vertical level information to generate the terrain following coordinates. The hybrid level specification is stored in <icon home>/hyb\_params/HYB\_PARAMS\_<nlev>. The **hydrostatic** model assumes to get **pressure based** coordinates, the **nonhydrostatic** model expects **height based** coordinates. For further information see <icon home>/hyb\_params/README.

## Discussion

Document last edited by *addyourname* on *insertdate* Document last edited by *S Gruber* on *08-01-2014*.