

EMEP MSC-W Model Training Course
24-26 April, 2013
at met.no, Oslo, Norway
Room: VIA, CIENS Building
Forskningsparken

Basic Exercises

1. Base Run:

A simple run for a particular year with the given meteorology and input data for the whole domain is called a Base run. The results will be written out into the \$HOMEDIR where the 'config_emep.nml' and 'modrun.sh' files are kept. The following set of output files will be created for a simulation of one full meteorological year 2010:

Base_fullrun.nc →

Annual average file in netCDF format. This file has only 1 timestep and the data is an average between the *\$startdate* and *\$enddate*.

Base_month.nc →

Monthly data in netCDF format. This file has 12 timesteps.

Base_day.nc →

Daily data in netCDF format. This has 366 timesteps. The first time step is written out at 06:00 hr on 1Jan and at the end of the run, one more record is written out into the 'day.nc' file, i.e., two records are written out on 31Dec. This is the reason to have 366 records in the daily file.

Base_hour.nc →

Hourly data in netCDF format. This has 8784 timesteps.

sites_2010.csv →

Station data in ascii format. The default code writes out all the variables into the output file. Stations are read from the file 'sites.dat' in the '/input' directory.

sondes_2010.csv →

Radiosonde data in ascii format for selected stations, levels and selected components. 'sondes.dat' file in the '/input' directory has the information about sondes stations. Levels and components are selected in 'My_Outputs_ml.f90'.

RunLog.out →

The log file with a summary of the run. This file contains information about the flags used in that particular run. Also it has a summary of the emission data and the mass balance check of three components – sulfur, nitrogen, carbon - at the end.

Timing.out →

Contains detailed information about the CPU time used for each process.

2. Run the model for a selected domain:

Select the domain from 'config_EMEPSTD.nml' file:

egs. For selecting a domain for the area 15W-45E and 30N-73N, edit the following in 'config_EMEPSTD.nml'. Use the program on <http://www.emep.int/grid/index.html> to convert to EMEP coordinates and convert them to model coordinates.

```
RUNDOMAIN = 120, 360, 24, 368, ! (15W-45E;30N-73N)
```

3. Making a run for a selected time period:

Select the time period 5-10 February 2008:

Choose the time period in the runscript. 'modrun.sh'

Choose the start date: startdate="2010 01 01"

Choose the end date: enddate="2010 01 01"

4. 3D concentrations:

My_Derived_ml.f90: egs. Piece of code for writing out surface concentration of 'CO' into daily outputs in units of ppb, and concentration of RDN in units of ugN into monthly output:

```
type(typ_s5i), public, parameter, dimension(75) :: &  
  OutputConcs = (/ &  
typ_s5i("CO", "ppb", D3,"AIR_CONCS", SPEC, D)&  
  ! ug/m3  
,typ_s5i("RDN", "ugN", D2,"AIR_CONCS", GROUP, D)/)
```

In this example, CO, and RDN are the components.

ppb, and ugN are the units of respective components.

D3 is the parameter that defines whether the output is written out into 3D or 2D fields. (D2 represents 2D fields).

5. Add more species to hourly netCDF output file:

My_Outputs_ml.f90:

nhourly_out, **nlevels_hourly**, **FREQ_HOURLY** are the variables that decide how many variable to write out, to how many levels and how frequent. Egs. Code follows for 3m O3 concentration in ppb. 600.0 means the upper limit of concentration.

```
integer, public, save  :: nhourly_out=1  ! No. outputs
integer, public, save  :: nlevels_hourly=20 ! No. of levels
integer, public, parameter :: FREQ_HOURLY = 1 ! 1 hours between outputs
```

```
hr_out = (/ &
Asc2D("o3_3m", "ADVppbv", IXADV_o3, &
ix1,ix2,iy1,iy2,1, "ppbv",PPBINV,600.0) &
```

6. Adding Species to sites and sondes .csv outputs:

My_Outputs_ml.f90:

sites.csv

```
NSITES_MAX = 99  & ! Max. no surface sites allowed
FREQ_SITE = 1    & ! Interval (hrs) between outputs
NADV_SITE = NSPEC_ADV & ! No. advected species (1 up to NSPEC_ADV)
NSHL_SITE = NSPEC_SHL & ! No. short-lived species
```

The default code writes out all components into '**sites.csv**' file. Variables 'NADV_SITE' and 'NADV_SHL' can be manipulated to select limited no. of variables.

sondes.csv

```
NSONDES_MAX = 99  & ! Max. no sondes allowed
NLEVELS_SONDE = 20 & ! No. k-levels (9 => 0--2500 m)
FREQ_SONDE = 1    & ! Interval (hrs) between outputs
NADV_SONDE = 2    & ! No. advected species
NSHL_SONDE = 3    & ! No. short-lived species
NXTRA_SONDE = 4   ! No. Misc. met. params
```

```

integer, public, parameter, dimension(NADV_SONDE) :: &
SONDE_ADV = (/ IXADV_O3, IXADV_NO2/)
integer, public, parameter, dimension(NSHL_SONDE) :: &
SONDE_SHL = (/ IXSHL_OH, IXSHL_OD, IXSHL_OP /)
character(len=10), public, parameter, dimension(NXTRA_SONDE) :: &
SONDE_XTRA= (/ "NOy ", "z_mid ", "p_mid ", "th  " /)

```

The code is very much self explanatory.

7. Adding more stations to sites and sondes file:

'sites.dat' and 'sondes.dat' files in the input directory contains the information about the stations and radiosonde locations to which data has to be written out. If data for more station locations are needed, add them to this file and the model will produce data for those locations as well. You need to know the geographical coordinates, latitude and longitude, of the stations.

8. Nesting:

The boundary conditions needed for EMEP MSC-W model is provided with the input data. The model can read Boundary conditions data from other models as well. These data has to be in netCDF format.

Different Nesting modes are:

- read the external BC data only,
- produce EMEP BC data from the simulation,
- read the external BC data and produce EMEP BC data,
- using the default EMEP BC data from the input data directory and write out EMEP BC at the end of the simulation,
- read the external BC data only in the beginning of the simulation,
- read external BC at the beginning of the simulation and write out EMEP BC at the end of the simulation.

These options are controlled by the '**MODE**' flag in the '*config_emep.nml*' file and the MODE options are:

config_emep.nml:

MODE=0 → do nothing;
MODE=1 → write at given NHOURLSAVE intervals in Nest_ml.f90 ;
MODE=2 → read;
MODE=3 → read and write at given NHOURLSAVE intervals ;
MODE=10 → write at end of run;
MODE=11 → read at start;
MODE=12 → read at start and write at end (BIC)

How to use External BC data (MODE=2):

Following is an example showing how to read 'MyBC.nc' as an external BC data. The model is reading in 3 variables – O3,PAN, and CO – from this data. See the section **&ExternalBICs_bc** .

Steps to follow:

Copy '**config_EMEPSTD.nml**' to '**config_NEST.nml**'

Edit 'config_NEST.nml':

&Nest_config

Choose the MODE. For egs.,

MODE = 2 for reading external BC
template_read_BC = 'MyBC.nc' (The name of your BC data file)
template_write = 'EMEP_OUT_YYYYMMDD.nc' (ONLY for **MODE=1, 3, 10 and 12**)

&ExternalBICs_config

USE_EXTERNAL_BIC = T,
EXTERNAL_BIC_NAME = 'MyBICScenario' (egs., IFS for BC data from ECMWF)
TOP_BC = T, (If you want to use Top BC also from your file)
filename_eta = 'filename.zaxis' (This is a text file containing the description of your vertical coordinate.)
&end

The checks for reading vertical levels in the BC data is done in the following order:

The variable '**hyam**' (hybrid 'a' coefficient at layer midpoint), exists → **eta coordinate**.
Level is indexed with '**k**' → **Sigma coordinate**

'**filename_eta**' exist → **eta coordinate** derived from 'vct'* information in *filename_eta*
Level indexed with '**lev**' and **no 'hyam' or 'filename_eta'** → **pressure coordinate**

Independent of the coordinates of the BC file, the BC levels will be interpolated into EMEP model levels. If the BC file level structure is not recognized, and there is no 'filename_eta' provided, the model will crash.

An example of the 'filename_eta' for EMEP model levels is given below. Here the 'vct' variable describes the model level boundaries in hybrid eta coordinate:

emep_eta.zaxis:

```
#
# zaxisID 0
#
zaxistype : surface
size      : 1
name      : sfc
longname  : surface
levels    : 0
#
# zaxisID 1
#
zaxistype = hybrid
size      : 20
name      : k
longname  : vertical sigma coordinates
units     : sigma_level
levels    : 0.0200 0.0600 0.1000 0.1425 0.1950 0.2635 0.3470
           0.4365 0.5215 0.5990 0.6695 0.7330 0.7895 0.8390
           0.8815 0.9170 0.9455 0.9670 0.9820 0.9940
vctsize   = 42
vct       =
           10000. 09600. 09200. 08800. 08350. 07750. 06980.
           06080. 05190. 04380. 03640. 02970. 02370. 01840.
           01380. 00990. 00670. 00420. 00240. 00120. 00000.
           0.0000 0.0400 0.0800 0.1200 0.1650 0.2250 0.3020
           0.3920 0.4810 0.5620 0.6360 0.7030 0.7630 0.8160
           0.8620 0.9010 0.9330 0.9580 0.9760 0.9880 1.0000
```

*'vct' is the vertical coordinate table describing the hybrid 'a' and 'b' values ('hyai' and 'hybi') at the layer interfaces in eta coordinate system. They must respect the following constraint:

$$\begin{aligned} \text{hyai}_t &= 0 & \text{hybi}_t &= 1 \\ \text{hyai}_0 &= P_t & \text{hybi}_0 &= 0 \end{aligned}$$

where $P_t \rightarrow$ Pressure at top.

&ExternalBICs_bc

```
description='MyBICScenario','Version name',3, ! name,version,size
map_bc=! emep,external,frac,wanted,found,IXADV,
        'O3'  , 'O3'  ,1.0,T,F,-1,
        'PAN' , 'PAN' ,1.0,T,F,-1,
        'CO'  , 'CO'  ,1.0,T,F,-1,
```

&end

Variables in **map_bc** mean the following:

- 1: Variable name in EMEP MSC-W model.
- 2: Variable name in the External BC data file.
- 3: External BC component to EMEP component fraction.
- 4: Is this component wanted or not. (should be T).
- 5: Was the BC variable found on the file (reset by the model)
- 6: Index of the advected model variable (reset by the model)

The fraction is helpful, when one has to map a variable that is explicitly not in EMEP model, but a fraction of that particular variable can be mapped to a matching variable in EMEP.

Caution should be given to the units of the variables in the External BC data file. The model tries to read in the variable together with its units. This is done in the module '**Units_ml.f90**'. The default units that EMEP can handle now are:

ugSm3, ugNm3, ugCm3, ugm3, ppb and mix_ratio (mol/mol).

If you have a BC data with different units, either convert them into one of the above mentioned units or add the respective conversion factor in the module 'Units_ml.f90'.

How to produce BC data from EMEP MSC-W model (MODE=1):

The time resolution of the BC data that has to be produced by EMEP model, is specified in the module 'Nest_ml.f90' .

Egs., **NHOURS** = 3, means the data will be written out at 3 hour interval.

One can choose the frequency in which the BC out files has to be created (template_write in config file), and also the domain for which the data has to be written out (sub domain part in config file). The default setup of the model creates BC data for the whole run domain.

Egs., **config_NEST.nml:**

```
MODE = 1.  
template_write = 'EMEP_OUT_YYYYMMDD.nc' (This option will write  
out a file everyday with 8 timesteps in each file with  
the option 'NHOURSAVE=3' in Nest_ml.f90).  
'EMEP_OUT_YYYYMM.nc' (will write out a file every  
month with no. of days of month * 8 timesteps in each  
file with 'NHOURSAVE=3' in Nest_ml.f90).  
  
!----- Sub domain for write modes 1,3.  
istart = 60,  
jstart = 11,  
iend = 107,  
jend = 58,
```

This will produce a 'EMEP_OUT' file for the domain 'i=60,107 and j=11,58'.

Please note that MODE=12 and MODE=13 will always write out fulldomain.

9. Source Receptor Calculations:

Source Receptor calculations are performed for calculating the contribution of emission from a particular country. This is controlled by 'femis.dat' files provided with the input data. This is a simple ascii file containing information about the country code and percentage of emission of each pollutant. Egs. Femis.dat file looks like :

Name	7	sox	nox	co	voc	nh3	pm25	pmco
28	0	1.0	1.0	1.0	1.0	1.0	1.0	1.0

where 'Name' represent the country code which is given in *Country_ml.f90*, 7 is the no. of pollutants, followed by pollutants in the rest of the columns. In this example the values 1.0 corresponding to each of the pollutants mean that the emissions of these pollutants are 100%. To reduce the emission of SOX by 20% from the country France, the 'femis.dat' file has to be:

Name	7	sox	nox	co	voc	nh3	pm25	pmco
8	0	.80	1.0	1.0	1.0	1.0	1.0	1.0

