

# EMEP MSC-W Model Training Course at met.no, Oslo, Norway

Room: Braavo, IT Building, MET  
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 2013:

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\_2013.csv and sites\_2013.nc→

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\_2013.csv and sondes\_2013.nc →

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\_emep.nml' file:

egs. For selecting a domain for the area 10E-55E and 30N-74N, edit the following in 'config\_emep.nml'. Use the program on <http://www.emep.int/grid/index.html> to convert to EMEP coordinates and convert them to model coordinates.

```
RUNDOMAIN = 40, 120, 24, 106, ! (10E-55E;30N-74N)
```

## 3. Making a run for a selected time period:

Select the time period 5-10 February 2013:

Choose the time period in config file in the subdirectory 'input', "config\_emep.nml"

Choose the start date: `startdate="2013 02 05"`

Choose the end date: `enddate="2013 02 10"`

## 4. 3D concentrations:

Edit the following part in "config\_emep.nml":

```
OutputConcs_config  
OutputConcs=
```

```
'SO2' , 'ug' , '3d', 'AIR_CONCS', 'SPEC' , 4,
```

SO2 concentration is written out into daily file (4)

## 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 outputs:

'Sites' and 'Sondes' ( ref. User guide for the details of these files) produce outputs in both ascii (.csv) and netCDF format.

The default Opensource code writes out 16 variables into the 'Sondes' file and all variables into the 'Sites' file. The module '**My\_Outputs\_ml.f90**' controls the selection of variables for these output files. Egs follows: Maximum number of sites allowed in this example is 99, but you can change this according to your need. All variables are written out to the 'sites' file in 1 hour interval in this example. '*NADV\_SITE*' and '*NADV\_SHL*' can be manipulated to select limited no. of variables.

### 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
```

### sondes.csv

Similar to sites, sondes outputs variables are also defined in the following way in 'My\_Outputs\_ml.f90'. The Opensource code write out 16 variables into 20 levels in 1 hour interval. The following example writes out 8 variables into the sondes output (2 advected species '*O<sub>3</sub>*' and '*NO<sub>2</sub>*' which are specified by '*SONDE\_ADV*' in the code, 3 short lived species '*OH*',

'OD', and 'OP' specified by 'SONDE\_SHL', and 3 met variables 'z\_mid', 'p\_mid' and 'th' specified by 'SONDE\_XTRA').

```
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 = 3 ! 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 = (/ "z_mid", "p_mid", "th" /)
```

## 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 ;  
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 produce BC data from EMEP MSC-W model (MODE=1):**

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.

An example to produce EMEP BC data at 3 hour interval is given below.

Egs., **&Nest\_config.nml:**

```
MODE = 1.  
NHOURLSAVE = 3, means the data will be written out at 3 hour interval.  
template_write = 'EMEP_OUT_YYYYMMDD.nc' (This option will write  
out a file everyday with 8 timesteps in each file).  
'EMEP_OUT_YYYYMM.nc' (will write out a file every  
month with no. of days of month * 8 timesteps in each).
```

```
!----- Sub domain for write modes 1,3.
```

```
istart = 60,  
jstart = 11,  
iend = 107,  
jend = 58,
```

**&end**

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.

## 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** in 'config\_emep.nml'.

Steps to follow:

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)
```

**&end**

**&ExternalBICs\_config**

```
USE_EXTERNAL_BIC      = T,
EXTERNAL_BIC_NAME     = 'MyBC',
EXTERNAL_BIC_VERSION = 'ECHAM5',
TOP_BC                = T,
filename_eta          = '$filename_eta',#filename_eta
```

**&end**

Do the mapping of the variables from 'MyBC.nc' to EMEP variables:

**&ExternalBICs\_bc**

```
! BC from ECHAM
description='ECHAM','MyBC',3, ! name,version,size
map_bc=! emep,external,frac,wanted,found,IXADV,
'O3'   , 'O3_VMR_inst'   ,1.0,T,F,-1,
'PAN'  , 'PAN_VMR_inst'  ,1.0,T,F,-1,
'CO'   , 'CO_VMR_inst'   ,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'.

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:

$$\begin{aligned}
 \text{hyai}_1 &= 0 & \text{hybi}_1 &= 1 \\
 \text{hyai}_0 &= P_t & \text{hybi}_0 &= 0
 \end{aligned}$$

where  $P_t \rightarrow$  Pressure at top.

In this table, the first 21 values in the 'vct' column are 'hyai' and the rest 21 are 'hybi' values in hPa.

## 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*, 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

```