BRAMS INSTALL MANUAL

Minimal way Installation

Rev 5.5

---

1. **Introduction**

The BRAMS minimal way instalation don't use some libraries like:

- GRIB2 (WGRIB2): used to read GFS/NCEP Model's analisys directly;
- NETCDF4 with CDF5: used to read GEOS/NASA Model's analisys directly;
- HDF5 : used to read some fixed data like NDVI

The minimal instalation may be used if you're reading GRADS data analisys generated from global model (GFS, GEOS, etc) or Brams native format (vfm) generated by dprep (geraDP) software. Even You only can read data fixed using bin or grads format.

2. **Pre requisites:**

   The BRAMS must be compiled using fortran/C compiler. The Model was tested using GNU (Gfortran&GCC), Intel compiler(Ifort,icc), Portland (pgf90, pgcc), IBM, NEC and Cray compilers.

   BRAMS works only in parallel mode. One can run the model using a single processor/core, but must do it using MPI with the MPIRUN command. If you don't have a MPI library installed we recommend download and install the last version of MPICH stable release.
   Also take care to choose the correct version to your OS.

   example:

```
cd ~/install
wget http://www.mpich.org/static/downloads/3.3.2/mpich-3.3.2.tar.gz
tar -zxvf mpich-3.3.2.tar.gz
cd mpich-3.3.2
./configure -disable-fast CFLAGS=-O2 FFLAGS=-O2 CXXFLAGS=-O2 FCFLAGS=-O2 --
prefix=/opt/mpich3
make
sudo make install
```

> NOTE: If Your system uses another compile than GNU use it in configure of MPICH.

3. **Getting the Model:**

The BRAMS is under version control using subversion. If You are INPE's internal colaborator You can get the BRAMS Versions right from projects site. If You are external user or colaborator You can get the stable or beta releases on INPE/CPTEC ftp server. To get the last beta realease please

```
cd ~/install
wget http://ftp.cptec.inpe.br/pesquisa/bramsrd/BRAMS/releases/beta/BRAMS_GEOS.tgz
```

4. **Unpacking & Compiling the Model:**

```
tar -xzvf BRAMS_GEOS.tgz
cd BRAMS_GEOS/build
./configure --program-prefix=BRAMS_5.4 --prefix=/home/user --enable-jules --with-chem=RELACS_TUV --with-aer=SIMPLE --with-fpcomp=/opt/mpich3/bin/mpif90 --with-cpcomp=/opt/mpich3/bin/mpicc --with-fcomp=gfortran --with-ccomp=gcc
make
make install
```

> Notes:
>
> - --prefix=/home/user - You must choose your > **user** instalation area;
> - In example above we use mpich3 and gfortran (GNU) > instalation. If You use another set of compiler and > mpi libs please, change it;
> - In example we use the mpi installed in /opt/mpich area. Point accordingly your distribution/instalation;
> - The binary executable will be installed (By make install) in **bin** folder created under --prefix you specified. This will be your "run" area.

5. **Getting fixed data and tables**

The model uses some fixed files like topography, vegetation, NDVI, etc. You must get the data and put it in a folder to be readed by model.

- **Getting table's data:**

```
cd /home/user/bin
wget http://ftp.cptec.inpe.br/pesquisa/bramsrd/BRAMS/files/bramsTablesRev2000.tgz
tar -xzvf bramsTablesRev2000.tgz
```

> Notes:
>
> - the table data size is 405MB
> - Remember Your user folder /home/**user**/bin

- **Getting the fixed data:**

```
wget http://ftp.cptec.inpe.br/pesquisa/bramsrd/BRAMS/files/datafix.tar
tar -xzvf datafix.tar
cd datafix
bunzip2 **/*.bz2
```

> Notes:
>
> - the table data size is 3.5GB and must be downloaded once;
> - All the data inside datafix is zipped using bzip2. Please, **bunziped all the data** inside all datafix sub-folders. The bunzip2 command may take a while!

6. **Getting the Initial and boundary conditions:**

   The model needs initial and boundary conditions to run. The data must be obtained in INPE/CPTECs ftp site. Before download the data we recommend to created a datain subfolder inside Your bin folder:

```
mkdir datain
  ```
  Get all data in [ftp data site]
(http://ftp.cptec.inpe.br/pesquisa/bramsrd/BRAMS/data/GRADS/) by choosing the d
ate of simulation. We recommend to get the date 15May2020 for test. You can get
 data for just one day of forecast or until 10 days.

Example 1 (getting all data - ctl and gra):

```bash
wget -nd -r -P ./datain -
A ctl,gra http://ftp.cptec.inpe.br/pesquisa/bramsrd/BRAMS/data/GRADS/20200515/
```

Example 2 (getting data for just 1 day- from 0h until 21h - ctl and gra):

```
wget -nd -r -P ./datain -
A IC20200515??.ctl http://ftp.cptec.inpe.br/pesquisa/bramsrd/BRAMS/data/GRADS/2
0200515/
wget -nd -r -P ./datain -
A IC20200515??.gra http://ftp.cptec.inpe.br/pesquisa/bramsrd/BRAMS/data/GRADS/2
0200515/
wget -nd -r -P ./datain -
A SM.GEOS.* http://ftp.cptec.inpe.br/pesquisa/bramsrd/BRAMS/data/GRADS/20200515
/
```

> Note:
> On both examples two kind of data are downloaded:
>
> - (a) - Initial & boundary conditions

- (b) - Soil Moisture conditions

7. **The tmp folder.**

   Brams uses Jules Models for surface. Jules needs a temporary directories for run. Please, make it inside your bin folder and export the tmp folder before run the model:

```
cd /home/user/bin
mkdir tmp
export TMPDIR=./tmp
```

8. **The NAMELIST (RAMSIN)**

The BRAMS uses by default a namelist called RAMSIN. We disposed a minimal RAMSIN to be modified by user, get it in ftp area:

```
wget http://ftp.cptec.inpe.br/pesquisa/bramsrd/BRAMS/files/RAMSIN_MINIMAL
```

Please, lets talk about some RAMSIN variables:

Variable	Values	Description
RUNTYPE	'MAKESFC','MAKEVFILE','INITIAL'	The BRAMS is executed in 3 phases: MAKESFC, MAKEVFILE and INITIAL. The first to phases prepares the model for run. They Creates fix data and data in the area of interest. You must run the MAKESFC and MAKEVFILE using just 1 processor (or core). The INITIAL phase is the integration in time. It reads the previous data from 2 previous phases and go forward. The INITIAL phase may be executed with the maximum of processors possible (it depends of domain configuration).
TIMMAX	Max Boundary Conditions time	define how long the forecast will be done
IMONTH1	1 to 12	Month of initial date
IDATE1	1 to 31	day of initial date
IYEAR1		Year of initial date
ITIME1	0000 to 2300	Hour and minute of initial date - all the time variables are used to define the time of model start

Variable	Values	Description
NGRIDS	1	Number of main+nested grids - Versions 5.0 and newest only admits 1 grid
NNXP	UD	Number of x gridpoints
NNYP	UD	Number of y gridpoints
NNZP	UD	Number of z gridpoints
NZG	UD	Number of soil layers
NZS	UD	Maximum number of snow layers
DELTAX	UD	X grid spacing [m]
DELTAY	UD	Y grid spacing [m]
DELTAZ	UD	initial Z grid spacing [m]
DZRAT	UD	Vertical grid stretch ratio
POLELAT	from -90 to +90	Latitude of pole point
POLELON	from -180 to +180	Longitude of pole point
CENTLAT	from -90 to +90	Latitude of grid center
CENTLON	from -180 to +180	Longitude of grid center
DTLONG	UD	Coarse grid long timestep [s]. This is one important value! This variable inform to model integration the delta T used to advance in time. Small numbers is best but the model will takes a lot of time to end. Large number may cause a model instability and crash. If You are using DYNCORE_FLAG = 0 (leapfrog method) pay attention in SSCOURN value printed in screen when models run, the value must be close to 1. If You are using DYNCORE_FLAG = 2 (Runge-Kupta) the value may be close to 4. When You run the model and it crash with blow up of courant number may be necessary decrease the DTLONG. The DTLONG depends of resolution of model. High resolutions needs small dtlong values.

Variable	Values	Description
FRQANL	UD	Time between outputs for forecast [s]. In some computer systems this time is relevant because it uses heavy I/O operations.You can test a model without I/O by setting IPOS = 0 and IOUTPUT = 0. By default we use only pos-processed grads file and turn off IOUTPUT. For IPOS = 2 the output will be the variables listed in $POST section (close to end of RAMSIN) and the files output are defined by GPREFIX.

> NOTES:
>
> - If You change one of the time or grid variables You must to run the 3 phases again.
> - The size defined by default is small enough to run in a ordinary laptop
> - In case the test with same size of the area/time/date you can only run the INITIAL phase.
> - IF You change the DELTAX or DELTAY the value of DTLONG must be change accordingly!

9. **Running the Model:**

   You must run the 3 phases of Model:

- [x] Edit RAMSIN and change RUNTYPE to 'MAKESFC'.
  Then submmit the run with only one processor (or core):

```
mpirun -np 1 brams-5.4 -f RAMSIN_MINIMAL
```

- [x] Edit RAMSIN and change RUNTYPE to 'MAKEVFILE'.
  Then submmit the run with only one processor (or core):

```
mpirun -np 1 brams-5.4 -f RAMSIN_MINIMAL
```

- [x] Edit RAMSIN and change RUNTYPE to 'INITIAL'.
  Then submmit the run with a lot of processors (or cores):

```
mpirun -np 40 brams-5.4 -f RAMSIN_MINIMAL
```

> NOTES:

- The number of cores You run depends of the machine architecture and the model's grid defined. If You use more cores than the model can support for one grid You must tune the grid. In general the problem accours when the number of grid's columns by core is less then halo of MPI communication. Small area with little colunms do not be used with many cores. The model will notice You when the problem is present.
- Pay attention to the output screen. Some errors may occur for a lot of reasons. Check them carefully in case of problems. Colors will used to show problems: green for notices, yellow for warnings and red for fatal problems.
- If You want to see the outputs during model runs just look inside the output GPREFIX folder to see the GRADS files generated.