

Set Up and Run WRF (*real* data)

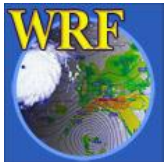
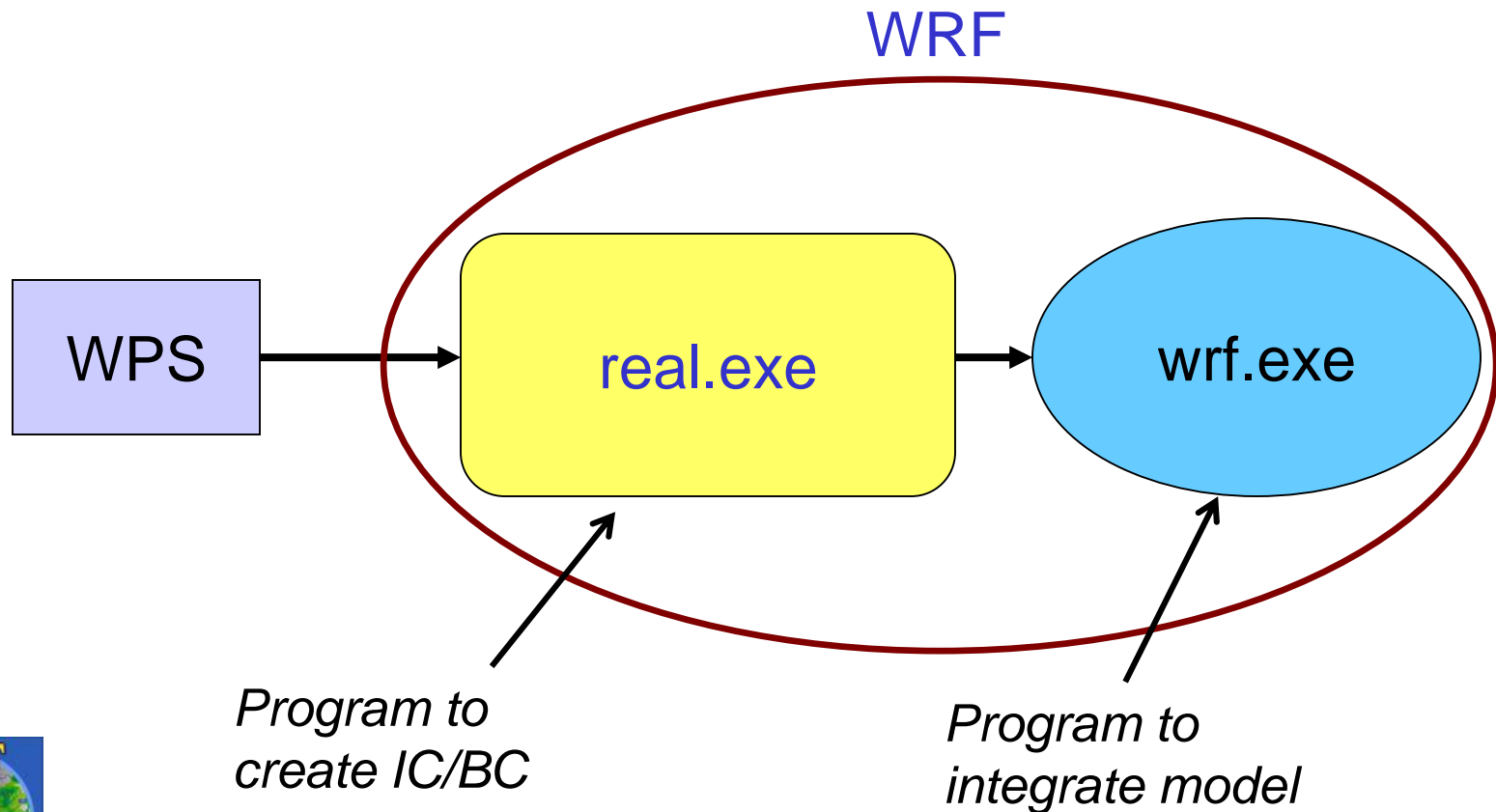
Wei Wang / Dave Gill

COAWST Workshop

25 February 2019



WRF System Flowchart



Outline

- Running WRF code
 - Things to check before you run..
 - Running **real-data** case
- Basic runtime options for a **single** domain run (*namelist*)
- Check output
- Simple trouble shooting



Before You Run ..

- Top directory is now **WRF/**
- Make sure appropriate executables are created in **WRF/main/** directory:
 - **real.exe** – executable to create IC/BC
 - **wrf.exe** – executable for model integration
 - **ndown.exe** – utility
 - **tc.exe** – utility routine for TC bogusing
- If you are working with real data, be sure that files for **a few time periods** from WPS are correctly generated:
 - **met_em.d01.***



WRF test case directories

You have these choices in **WRF/test/**
(choices made at compile time. E.g. *compile em_real*):

`em_real`



Steps to Run

1. cd to *run/* or the *test/em_real/* directory
2. Move or link WPS output files to the directory for real-data cases
3. Edit *namelist.input* file for the appropriate grid dimensions and times of the case
4. Run a initialization program (*real.exe*)
5. Run model executable, *wrf.exe*



WRF/run directory

README.namelist

LANDUSE.TBL

GENPARM.TBL

SOILPARM.TBL

VEGPARM.TBL

URBPARM.TBL

RRTM_DATA

RRTMG_SW_DATA

RRTMG_LW_DATA

CAM_ABS_DATA

CAM_AEROPT_DATA

ozone.formatted

ozone_lat.formatted

ozone_plev.formatted

aerosol.formatted

aerosol_lat.formatted

aerosol_lon.formatted

aerosol_plev.formatted

gribmap.txt

grib2map.tbl

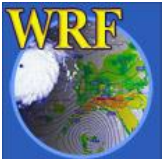
.... (a total of 60 files)

} *description of namelists*

These are model physics data files: they are used to either initialize physics variables, or make physics computation faster

** Some of these files are text files, hence editable*

} *for grib IO*



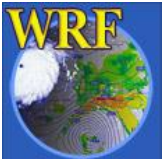
WRF/run directory after compile

```
LANDUSE.TBL
SOILPARM.TBL
VEGPARM.TBL
GENPARM.TBL
URBPARM.TBL
RRTM_DATA
RRTMG_SW_DATA
RRTMG_LW_DATA
ozone.formatted
ozone_lat.formatted
ozone_plev.formatted
```

*An example after
em_real case
compile*

...

```
namelist.input - copied from ../test/em_real/namelist.input
real.exe -> ../main/real.exe
wrf.exe -> ../main/wrf.exe
ndown.exe -> ../main/ndown.exe
.... (a few more)
```



Running a Real-Data Case



Running a Real-Data Case

- If you have compiled the *em_real* case, you should have:

real.exe - *real data initialization program*

wrf.exe - *model executable*

ndown.exe - *program for doing one-way nesting*

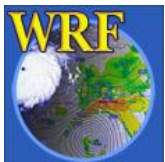
tc.exe - *program for TC bogusing*

- These executables are linked to:

WRF/run

and

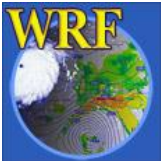
WRF/test/*em_real*



➔ One can go to either directory to run.

WRF/test/em_real directory

```
LANDUSE.TBL -> ../../run/LANDUSE.TBL
GENPARAM.TBL -> ../../run/GENPARAM.TBL
SOILPARAM.TBL -> ../../run/SOILPARAM.TBL
VEGPARM.TBL -> ../../run/VEGPARM.TBL
URBPARM.TBL -> ../../run/URBPARM.TBL
RRTM_DATA -> ../../run/RRTM_DATA
RRTMG_SW_DATA -> ../../run/RRTMG_SW_DATA
RRTMG_LW_DATA -> ../../run/RRTMG_LW_DATA
ozone.formatted -> ../../run/ozone.formatted
ozone_lat.formatted -> ../../run/ozone_lat.formatted
ozone_plev.formatted -> ../../run/ozone_plev.formatted
...
namelist.input - editing required
real.exe -> ../../main/real.exe
wrf.exe -> ../../main/wrf.exe
ndown.exe -> ../../main/ndown.exe
.... (many more)
```

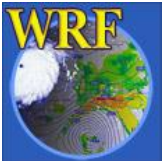


Running a Real-data Case

- One must successfully run WPS to prepare data required, and create `met_em.*` files for multiple time periods for initial and boundary conditions
- Move or link WPS/metgrid output files to the run directory:

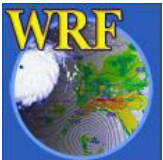
```
cd test/em_real
```

```
ln -s ../ ../WPS/met_em.d01.* .
```



Running a Real-data Case

- Edit `namelist.input` file for runtime options (*at minimum*, one must edit `&time_control` for start, end and integration times, and `&domains` for grid dimensions)
- Run the real-data initialization program:
`./real.exe` if compiled serially / SMP, or
`mpirun -np N ./real.exe` for a MPI job
where `N` is the number of processors requested.



Running a Real-data Case

- Successfully running `real.exe` will create model initial and boundary files:

`wrfinput_d01`

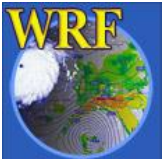
`wrfbdy_d01`

*Single time level
data at model's
start time*

*N-1 time-level data for
lateral boundaries, and
only for domain 1*

N: the number of time periods processed

```
ncdump -v Times wrfbdy_d01
```



Running a Real-data Case

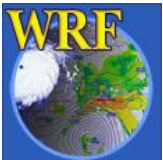
- Typing `'ncdump -v Times wrfbdy_d01'` will give you, for a 24 hour period, 6 hourly data interval:
.. a bunch of prints and at the end:

data:

Times =

```
"2005-08-28_00:00:00",  
"2005-08-28_06:00:00",  
"2005-08-28_12:00:00",  
"2005-08-28_18:00:00" ;
```

* BC data consists of values at the start of the time interval and rate of change in the time interval.



Running a Real-data Case

- Run the model executable by typing:

```
./wrf.exe >& wrf.out &
```

or

```
mpirun -np N ./wrf.exe &
```

- Successfully running the model will create model *history* file:

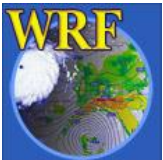
```
wrfout_d01_2005-08-28_00:00:00
```

Based on start date set in namelist

and a *restart* file if `restart_interval` is set to a time within the range of the forecast time:

```
wrfrst_d01_2005-08-28_12:00:00
```

Exact time at a restart



Running a Real Data Case

wrfout_d01_2005-08-28_00:00:00

Based on start date set in namelist

start_year	= 2008,	2008,	2008,
start_month	= 08,	08,	08,
start_day	= 28,	28,	28,
start_hour	= 00,	00,	00,
start_minute	= 00,	00,	00,
start_second	= 00,	00,	00,
end_year	= 2008,	2008,	2008,
end_month	= 08,	08,	08,
end_day	= 29,	29,	29,
end_hour	= 00,	00,	00,
end_minute	= 00,	00,	00,
end_second	= 00,	00,	00,



Basic namelist Options



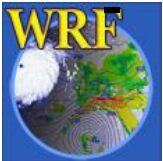
What is a namelist?

- A Fortran namelist contains a list of *runtime* options for the code to read in during its execution. Use of a namelist allows one to change runtime configuration without the need to recompile the source code.
- Fortran 90 namelist has very specific format, so edit with care:

```
&namelist-record - start  
/                - end
```

- As a general rule:
 - Multiple columns: domain dependent
 - Single column: value valid for all domains

A namelist file may contain a number of records



namelist record `&time_control`

```
run_days           = 0,  
run_hours          = 24,  
run_minutes        = 0,  
run_seconds        = 0,  
start_year         = 2000, 2000, 2000,  
start_month        = 01, 01, 01,  
start_day          = 24, 24, 24,  
start_hour         = 12, 12, 12,  
start_minute       = 00, 00, 00,  
start_second       = 00, 00, 00,  
end_year           = 2000, 2000, 2000,  
end_month          = 01, 01, 01,  
end_day            = 25, 25, 25,  
end_hour           = 12, 12, 12,  
end_minute         = 00, 00, 00,  
end_second         = 00, 00, 00,  
interval_seconds   = 21600  
history_interval   = 180, 60, 60,  
frames_per_outfile = 1000, 1000, 1000,  
restart_interval   = 360,  
restart            = .true.,
```

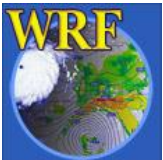
domain 1 option

for nests



Notes on `&time_control`

- `run_*` time variables:
 - Model simulation length: `wrf.exe` and domain 1 only
- `start_*` and `end_*` time variables:
 - Program `real` will use WPS output between these times to produce lateral (and lower) boundary file
 - They can also be used to specify the start and end of simulation times for the coarse grid if `run_*` variables are not set (or set to 0)



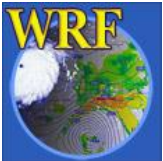
Notes on `&time_control`

- *interval_seconds*:
 - Time interval between WPS output times, and lateral BC (and lower BC) update frequency
- *history_interval*:
 - Time interval in minutes when a history output is written (*note* output is instantaneous)
 - If the `time_step` cannot be evenly divided by `history_interval`, then nearest time step output is used
 - The time stamp in a history file name is the time when the history file is first written, and multiple time periods may be written in one file. e.g. a history file for domain 1 that is first written for 1200 UTC Jan 24 2000 is
`wrfout_d01_2000-01-24_12:00:00`



Notes on `&time_control`

- *frames_per_outfile*:
 - Number of history times written to one file
- *restart_interval*:
 - Time interval in minutes when a restart file is written
 - By default, restart file is not written at hour 0
 - A restart file contains only one time level data, and its valid time is in its file name, e.g. a restart file for domain 1 valid for 0000 UTC Jan 25 2000 is
`wrfrst_d01_2000-01-25_00:00:00`
- *restart*:
 - whether this is a restart run



Notes on `&time_control`

Example 1: all output times are in a single file

```
history_interval      = 180, 60, 60,  
frames_per_outfile   = 1000, 1000, 1000,  
wrfout_d01_2000-01-24_12:00:00
```

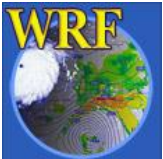
Example 2: each output file only contains a single time

```
history_interval      = 180, 60, 60,  
frames_per_outfile   = 1, 1, 1,  
wrfout_d01_2000-01-24_12:00:00  
wrfout_d01_2000-01-24_15:00:00  
wrfout_d01_2000-01-24_18:00:00
```



Notes on *restart*

- What is a *restart* run?
 - A restart run is a continuation of a model run
- How to do a *restart* run:
 - In the first run, set *restart_interval* to a value that is within the model integration time
 - A restart file will be created. e.g.
`wrfirst_d01_2000-01-25_00:00:00`
- When doing a restart run:
 - Set *restart* = .true.,
 - Set start time to restart time
 - Set run_* to be the hours remaining in the run



&time_control

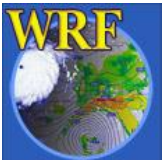
```
io_form_history      = 2,  
io_form_restart     = 2,  
io_form_input       = 2,  
io_form_boundary    = 2,
```

IO format options:

- = 1, binary
- = 2, netcdf (most common)
- = 4, PHDF5
- = 5, Grib 1
- =10, Grib 2
- =11, pnetCDF

For large files:

```
io_form_restart = 102 :  
write output in patch  
sizes: fast for large grid  
and useful for restart file
```



namelist record **&domains**

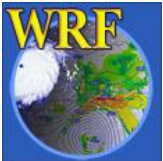
```
time_step           = 180
time_step_fract_num = 0,
time_step_fract_den = 1,
max_dom             = 1,
e_we                = 74, 112, 94,
e_sn                = 61, 97, 91,
e_vert              = 33, 28, 28,
num_metgrid_levels  = 32,
num_metgrid_soil_levels = 4
dx                  = 30000, 10000, 3333,
dy                  = 30000, 10000, 3333,
eta_levels           = 1.0, 0.996, 0.99, 0.98, ... 0.0
p_top_requested      = 5000,
```

nest
options



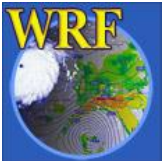
Notes on `&domains`

- `time_step, time_step_fract_num, time_step_frac_den`:
 - Time step for model integration in seconds
 - Fractional time step specified in separate integers of numerator and denominator
 - Typically 5 to $6 \times DX$ (DX is grid distance in km)
- `e_we, e_sn, e_vert`:
 - Model grid dimensions (staggered) in X, Y and Z directions, need to match those defined in `geogrid` program
- `num_metgrid_levels`:
 - Number of `metgrid` (input) data levels
- `num_metgrid_soil_levels`:
 - Number of soil data levels in the input data
- ➔ Both can be found by typing `ncdump -h met_em.d01.<date> | more`
- `dx, dy`:
 - grid distance: in meters



Notes on `&domains`

- *p_top_requested*:
 - Pressure value at the model top
 - Constrained by the available data from WPS
 - Default is 5000 Pa (recommended as lowest model top)
- *eta_levels*:
 - Specify your own model levels from 1.0 to 0.0
 - If not specified, program *real* will calculate a set of levels
 - V4 has a new and better way to compute the levels
 - Use a minimum of 33 or more levels with 5000 Pa model top to limit vertical grid distance < 1 km. Use more vertical levels when decreasing horizontal grid sizes.



namelist record `&bdy_control`

`spec_bdy_width`

`spec_zone`

`relax_zone`

`specified`

`nested`

typical

optional

`= 5, (10)`

`= 1, (1)`

`= 4, (9)`

`= .true.,`

`= .false.,`

do not change

May change `relax_zone`
and `spec_bdy_width`
(`spec_zone + relax_zone`
`= spec_bdy_width`)

* Wider boundary zone may work
better for coarser driving data



Other namelists

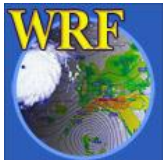
&physics:

- Model physics options

&dynamics:

- Damping, diffusion options
- Advection options
- In 4.0, the hybrid vertical coordinate option is the default. Turn it off by setting the following for *real* and *wrf*:

```
hybrid_opt          = 0
```



Where do I start?

- Always start with a *namelist* template provided in a test case directory, whether it is an ideal case, or a real data case.
 - A number of namelist templates are provided in *test/test_<case>/* directories

For example: in *test/em_real/*, there are

`namelist.input.4km` ~ 4 km grid size

`namelist.input.jun01` ~ 10 km grid size

`namelist.input.jan00` ~ 30 km grid size



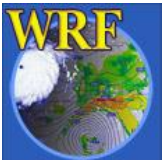
Where do I start?

- For different applications, please refer to p5-38 to 5-40 of the ARW User's Guide:
 - 2 or 4 km microphysics-only runs
 - 20 – 30 km, 2 – 3 day runs
 - Antarctic region
 - Tropical storm forecasting
 - Regional climate
 - Try physics suites (since V3.9)



Where do I start?

- Use document to guide the modification of the namelist values:
 - `run/README.namelist`
 - `test/em_real/examples.namelist`
 - User's Guide, Chapter 5 (online version has the latest)
 - Full list of namelists and their default values can be found in Registry files: [Registry.EM_COMMON](#), `registry.io_boilerplate` (for IO options) and other registry files - look for character string '*namelist*'

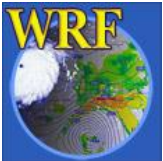


To run a job in a different directory..

- Directories *run/* and *test_<case>/* are convenient places to run, but it does not have to be.
- Copy or link the content of these directories to another directory, including **physics data** files, wrf **input** and **boundary** files, wrf **namelist** and **executables**, and you should be able to run a job anywhere on your system.



Check Output



Output After a Model Run

- Standard out/error files:
`wrf.out`, or `rs1.*` files
- Model history file(s):
`wrfout_d01_<date>`
- Model restart file(s), optional
`wrfirst_d01_<date>`



Output from a multi-processor run

The standard out and error will go to the following files for a MPI run:

```
mpirun -np 4 ./wrf.exe →
```

```
rsl.out.0000
```

```
rsl.error.0000
```

```
rsl.out.0001
```

```
rsl.error.0001
```

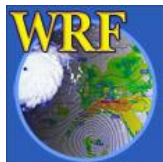
```
rsl.out.0002
```

```
rsl.error.0002
```

```
rsl.out.0003
```

```
rsl.error.0003
```

There is one pair of files for each processor requested



What to Look for in a standard out File?

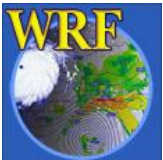
Check run log file by typing

```
tail wrf.out, or
```

```
tail rsl.out.0000
```

You should see the following if the job is successfully completed:

```
wrf: SUCCESS COMPLETE WRF
```



How to Check Model History File?

- Use **ncdump** :

```
ncdump -v Times wrfout_d01_<date>
```

to check output times. Or

```
ncdump -v U wrfout_d01_<date>
```

to check a particular variable (U)

- Use **ncview** (great tool!)
- Use post-processing tools (see talks later)



What is in a *wrf.out* or *rsl* file?

- Model version, decomposition info:

```
Ntasks in X          2, ntasks in Y          4
WRF V4.0 MODEL
```

- Time taken to compute one model step:

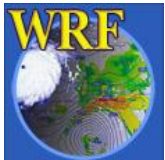
```
Timing for main: time 2000-01-24_20:03:00 on domain 1: 0.89475 elapsed seconds
Timing for main: time 2000-01-24_20:06:00 on domain 1: 0.09011 elapsed seconds
Timing for main: time 2000-01-24_20:09:00 on domain 1: 0.08634 elapsed seconds
Timing for main: time 2000-01-24_20:12:00 on domain 1: 0.09004 elapsed seconds
```

- Time taken to write history and restart file:

```
Timing for Writing wrfout_d01_2000-01-25_00:00:00 for domain 1: 0.07091 elapsed seconds
```

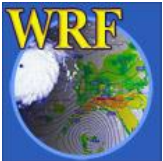
- Any model error prints:

```
5 points exceeded cfl=2 in domain 1 at time 4.200000 MAX AT i,j,k: 123 48 3
cfl,w,d(eta)= 4.165821
```



→ An indication the model has become numerically unstable

Simple Trouble Shooting



Often-seen runtime problems

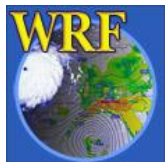
- module_configure: initial_config: error reading

namelist: &dynamics

> Typos or erroneous namelist variables exist in namelist record *&dynamics* in *namelist.input* file

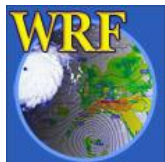
- input_wrf.F: SIZE MISMATCH: namelist
ide,jde,num_metgrid_levels= 70 61 27 ; input
data ide,jde,num_metgrid_levels= 74 61 27

> Grid dimensions in error



Often-seen runtime problems

- **Segmentation fault** (core dumped)
 - > Often typing **'unlimit'** or **'ulimit -s unlimited'** or equivalent can help when this happens quickly in a run, and on a small computer
- If you do: `grep cfl rsl.error.*` and see
121 points **exceeded cfl=2** in domain 1 at time
4.200000 MAX AT i,j,k: 123 48 3 cfl,w,d(eta)=
4.165821
 - > Model becomes unstable due to various reasons. If it happens soon after the start time, check input data, and/or reduce time step.



References

- Information on compiling and running WRF, and a more extensive list of namelist options and their definition / explanations can be found in the [User's Guide, Chapter 5](#)

