

WRF: *More Runtime Options*

Wei Wang

July 2023

Mesoscale and Microscale Meteorology Laboratory, NCAR



More options

- Some useful *runtime* options:
 - Vertical interpolation options (program *real.exe*, [&domains](#))
 - Base state parameters (*real.exe*, [&dynamics](#))
 - Defining vertical model levels (*real.exe*, [&domains](#))
 - IO options ([&time_control](#))
 - Physics suites ([&physics](#))
 - Options for long simulations ([&physics](#))
 - Adaptive-time step ([&domains](#))
 - Digital filter ([&dfi_control](#))
 - Stochastic parameterization schemes ([&stoch](#))
 - Tracer ([&dynamics](#)) / trajectory ([&physics](#), [&domains](#))
 - Optional output (various)
 - IO quilting ([&namelist_quilt](#))
- Time series output (surface and profile)



namelist.input

general namelist
records:

`&time_control`
`&domains`
`&physics`
`&dynamics`
`&bdy_control`
`&namelist_quilt`

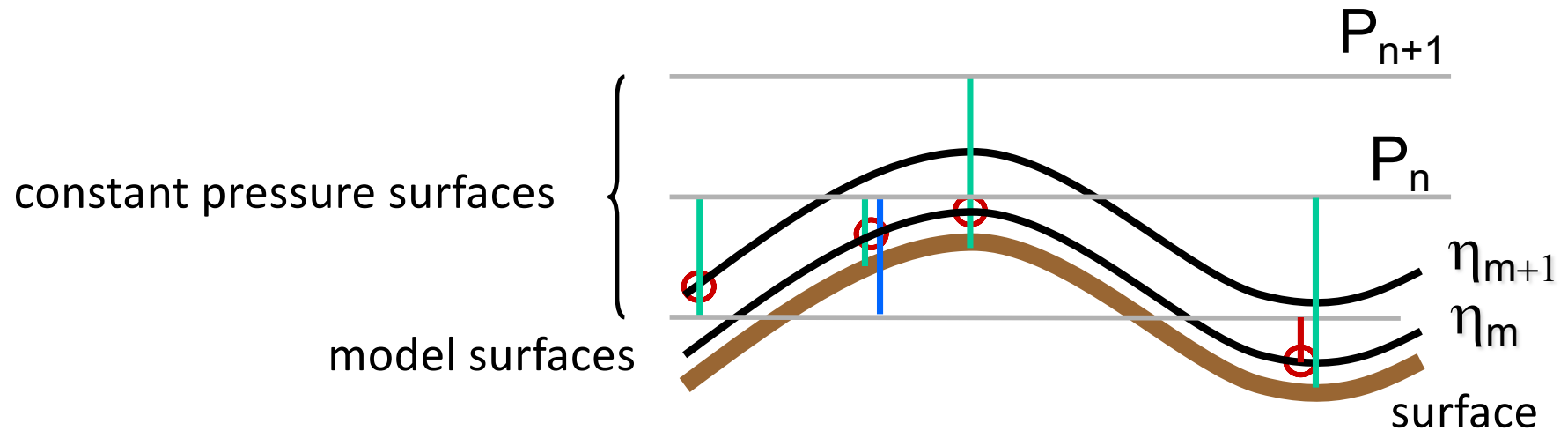
specialized namelist
records:

`&dfi_control`
`&fdda`
`&stoch`
`&diags`
`&scm`
`&tc`
`&noah_mp`



Look for these in test/em_real/**examples.namelist**

Vertical interpolation options (1)



Program **real.exe**, &domains:

use_surface: whether to use surface level data (default is true)

lagrange_order: linear, quadratic (default) or cubic



Vertical interpolation options (2)

Program **real** only, **&domains**:

use_levels_below_ground: whether to use data below the ground, T/F (default T)

lowest_lev_from_sfc: whether surface data is used to fill the lowest model level values, T/F (default F)

force_sfc_in_vinterp: number of levels to use surface data, default is 1

extrap_type: how to do extrapolation: 1 - use 2 lowest levels; 2 – constant (default)

t_extrap_type : extrapolation option for temperature: 1 - isothermal; 2 - 6.5 K/km (default); 3 - adiabatic

Look for these in examples.namelist



Base State Parameters

The following could be varied (program **real**, &**dynamics**):

base_temp¹

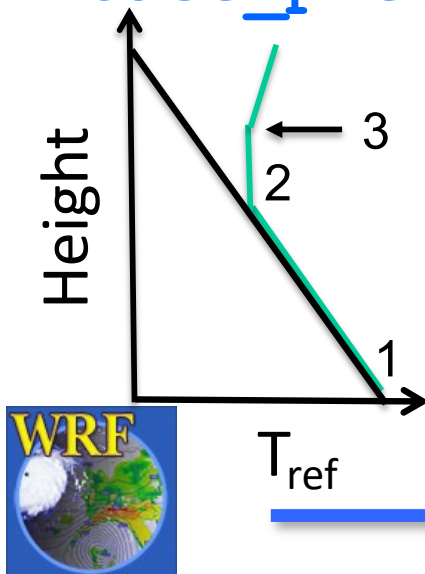
Base state surface temperature
(default 290 K)

iso_temp²

Base state stratosphere temperature
(default 200 K)

base_pres_strat³

Pressure at which the stratosphere
temperature lapse rate changes
(default 0 hPa)



The purpose is to minimize perturbation fields to improve solution accuracy when discretized.

Defining Vertical Levels

Program `real.exe`:

`&domains`

`auto_levels_opt = 2` (default since v4.0)

`dzstretch_s = 1.1 ~ 1.3`

`dzstretch_z = 1.02 ~ 1.1`

`dzbot = 20 ~ 50`

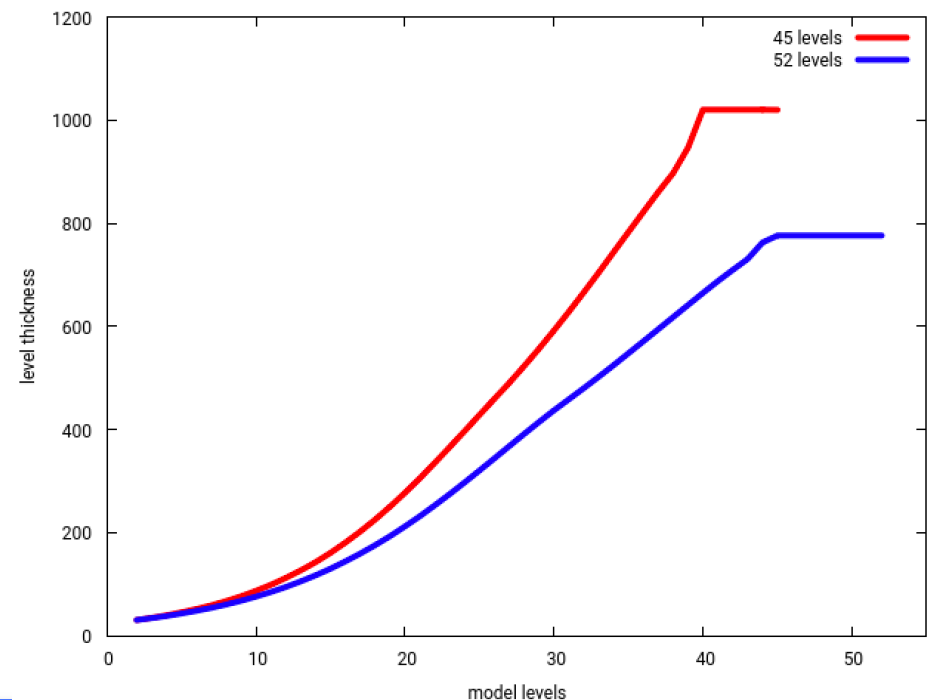
`max_dz = < 1000`

Example on the right:

`p_top_requested = 5000`

`e_vert = 52 (45)`

*See WRF Initialization/Setting
Model Vertical Levels in User's
Guide*



IO Control (1)

History output control in `&time_control`

<code>history_interval:</code>	used often, unit in <u>minutes</u>
<code>history_interval_h:</code>	history output interval in hours
<code>history_interval_s:</code>	history output interval in seconds
<code>history_begin_h:</code>	history output beginning time in hours
<code>history_begin_d:</code>	history output beginning time in days

Look for the list in

`Registry/registry.io_boilerplate`



IO Control (2)

Specify input and output files explicitly in
&time_control

```
auxinput1_inname = "/mydirectory/met_em.d<domain>.<date>"
```

- explicitly specify input file (it name and directory)

```
history_outname = "/mydirectory/wrfout_d<domain>_<date>"
```

- explicitly specify history output file (its name and directory)

Look for these in

Registry/registry.io_boilerplate



IO Control (3)

Optional output in `&time_control`

1. Change `Registry.EM_COMMON` and recompile:

```
state integer rainc ij misc 1 - h03 "RAINC"  
    "" "ACCUMULATED TOTAL CUMULUS PRECIPITATION"  
state integer rainnc ij misc 1 - h03 "RAINC"  
    "" "ACCUMULATED TOTAL GRID SCALE PRECIPITATION"
```

2. Edit `namelist.input` to output these variables:

```
auxhist3_outname = "rainfall_d<domain>"  
auxhist3_interval = 10, 10,  
frames_per_auxhist3 = 1000, 1000,  
io_form_auxhist3 = 2
```

Good for production runs



IO Control (4)

There is an alternative way to add/remove output fields at **runtime** (state variables in Registry only)

1. namelists in **&time_control**:

```
iofields_filename(max_dom) = 'my_output.txt',  
ignore_iofields_warning = .true.
```

2. prepare a text file ('my_output.txt') to select io fields:

```
+:h:3:rainc,rainnc    ← syntax in the file
```

3. set other namelists under **&time_control**:

```
auxhist3_outname = "rainfall_d<domain>"  
auxhist3_interval = 10, 10,  
frames_per_auxhist3 = 1000, 1000,  
io_form_auxhist3 = 2
```

Good for development runs

See 'WRF Output/**Run-Time IO**' section in User's Guide



Use of physics suite

Since 3.9, physics can be selected as a suite. These represent well-tested physics (**&physics**).

`physics_suite = 'tropical'`



```
mp_physics = 6, 6,  
cu_physics = 16, 16,  
ra_lw_physics = 4, 4,  
ra_sw_physics = 4, 4,  
bl_pbl_physics = 1, 1,  
sf_sfclay_physics = 91, 91,  
sf_surface_physics = 2, 2,
```

`physics_suite = 'CONUS'`



```
mp_physics = 8, 8,  
cu_physics = 6, 6,  
ra_lw_physics = 4, 4,  
ra_sw_physics = 4, 4,  
bl_pbl_physics = 2, 2,  
sf_sfclay_physics = 2, 2,  
sf_surface_physics = 2, 2,
```

* *Note other configuration choices can have an impact on model results*



Use of physics suite

To turn an option off for a particular domain:

```
physics_suite = 'tropical'
```



```
cu_physics = -1, 0,
```

-1: using suite option

To overwrite one or more with other options:

```
physics_suite = 'CONUS'
```



```
cu_physics = 16, 16,  
bl_pbl_physics = 1, 1,  
sf_sfclay_physics = 1, 1,
```



Options for long simulations (1)

Update control for lower boundary fields: allow SST, seaice, monthly vegetation fraction and albedo to be updated regularly during a model run:

sst_update: 0 – no update

1 – update all above fields (in **&physics**)

Set before running **real.exe**, and this will create additional output files: **wrflowinp_d01, wrflowinp_d02, ..**

Other namelists required in **&time_control**:

auxinput4_inname = “wrflowinp_d<domain>”

auxinput4_interval = 360, 360,

io_form_auxinput4 = 2 (netCDF)

See ‘Run-time Capabilities/**SST Update**’ in User’s Guide



Options for long simulations (2) (&physics)

sst_skin

diurnal water temp update

tmn_update

deep soil temp update, used with lagday

lagday

averaging time in days

bucket_mm

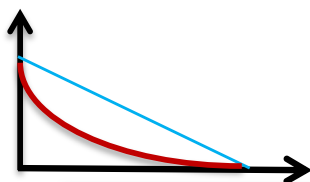
bucket reset value for rainfall
(e.g. `rainc=i_rainc*bucket_mm+rainc`)

bucket_j

bucket reset value for radiation fluxes

spec_exp

exponential multiplier for boundary zone ramping
(set in real, &bdy_control). Usually used with
wider boundary zone



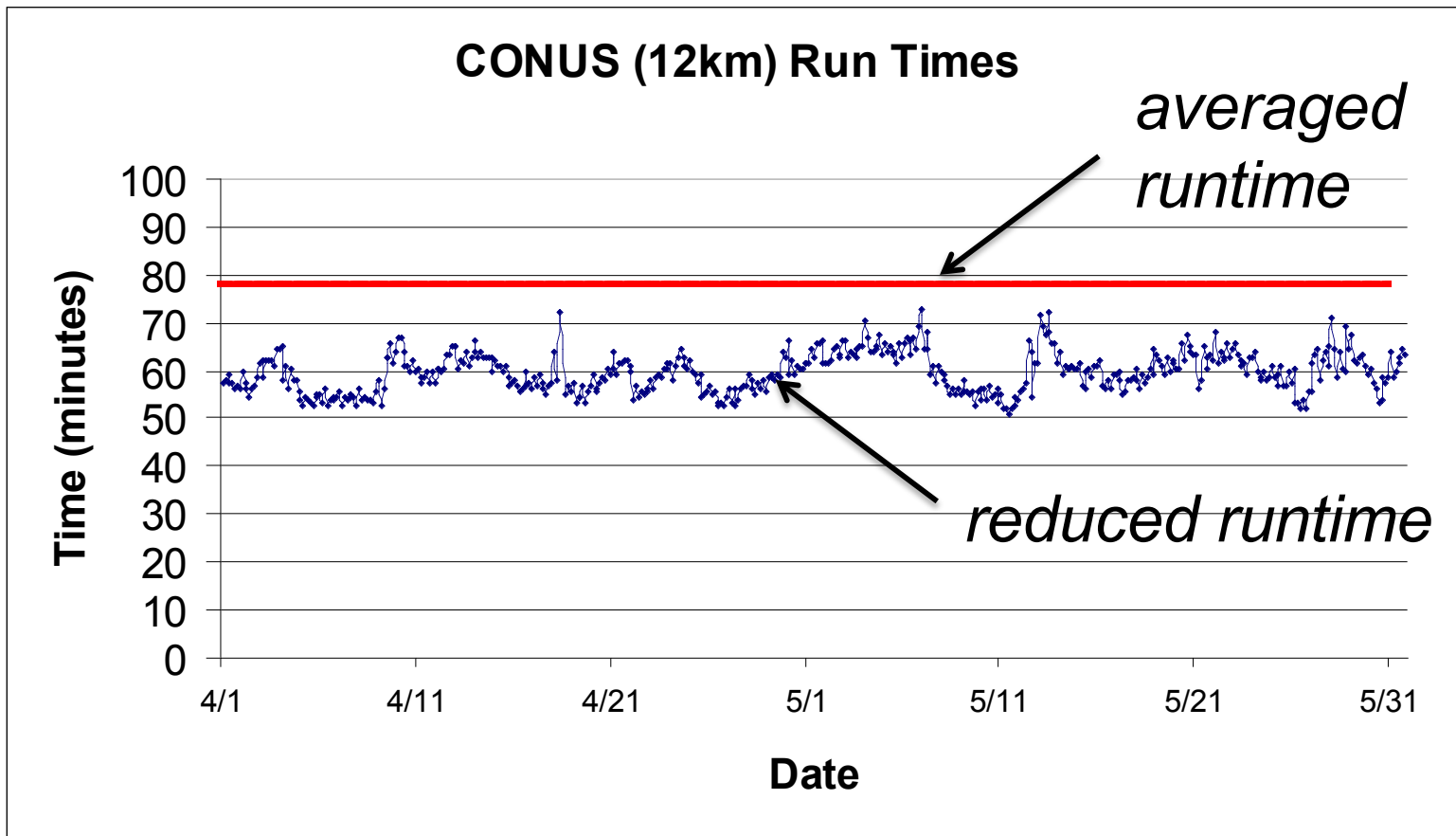
Adaptive time steps (1)

- Adaptive-time-step is a way to maximize the model time step while keeping the model numerically stable.
- Good to use for real-time run.
- May not work in combination with other options.

Also see ‘Run-time Capabilities/**Adaptive Time Stepping**’ section in User’s Guide



Adaptive time steps (2): an example



On average, forecasts finish in 60 min (50-73min)
as compared to 79 min standard runtime



Adaptive time steps (3)

Namelist control: **&domains** USE WITH CARE

`use_adaptive_time_step`

ture or false

`step_to_output_time`

whether to write at exact history output times

`target_cfl`

maximum cfl allowed (1.2)

`max_step_increase_pct`

percentage of time step increase each time; set to 5, 51, 51 (larger value for nest)

`starting_time_step`

in seconds; e.g. set to $4 \cdot DX$

`max_time_step`

in seconds; e.g. set to $8 \cdot DX$

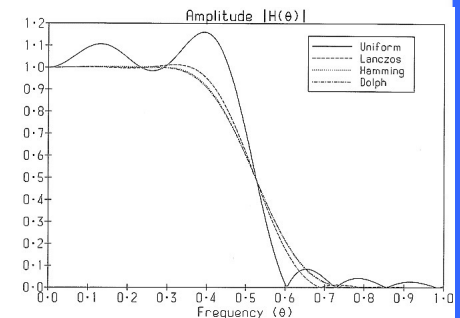
`min_time_step`

in seconds; e.g. set to $4 \cdot DX$



Digital Filter Initialization (DFI) (1)

- DFI is a way to use a low-pass filter to remove noise in model initial conditions
- Imbalances in model IC
 - May be introduced by interpolation, different topography, or by objective analysis, and data assimilation
 - May generate spurious gravity waves in the early simulation hours, which could cause erroneous vertical motion and precipitation, numerical instability and degrade subsequent data assimilation
- Useful for short-range model runs (1-6 hours)



Digital filter initialization (2)

Using DFI

- can construct consistent model fields which do not exist in the initial conditions, e.g. vertical motion, cloud variables
- may reduce the spin-up problem in early simulation hours
- Useful for short-range (1-6 h) forecasts and cycling with data assimilation

DFI is done after program **real**, or data-assimilation step

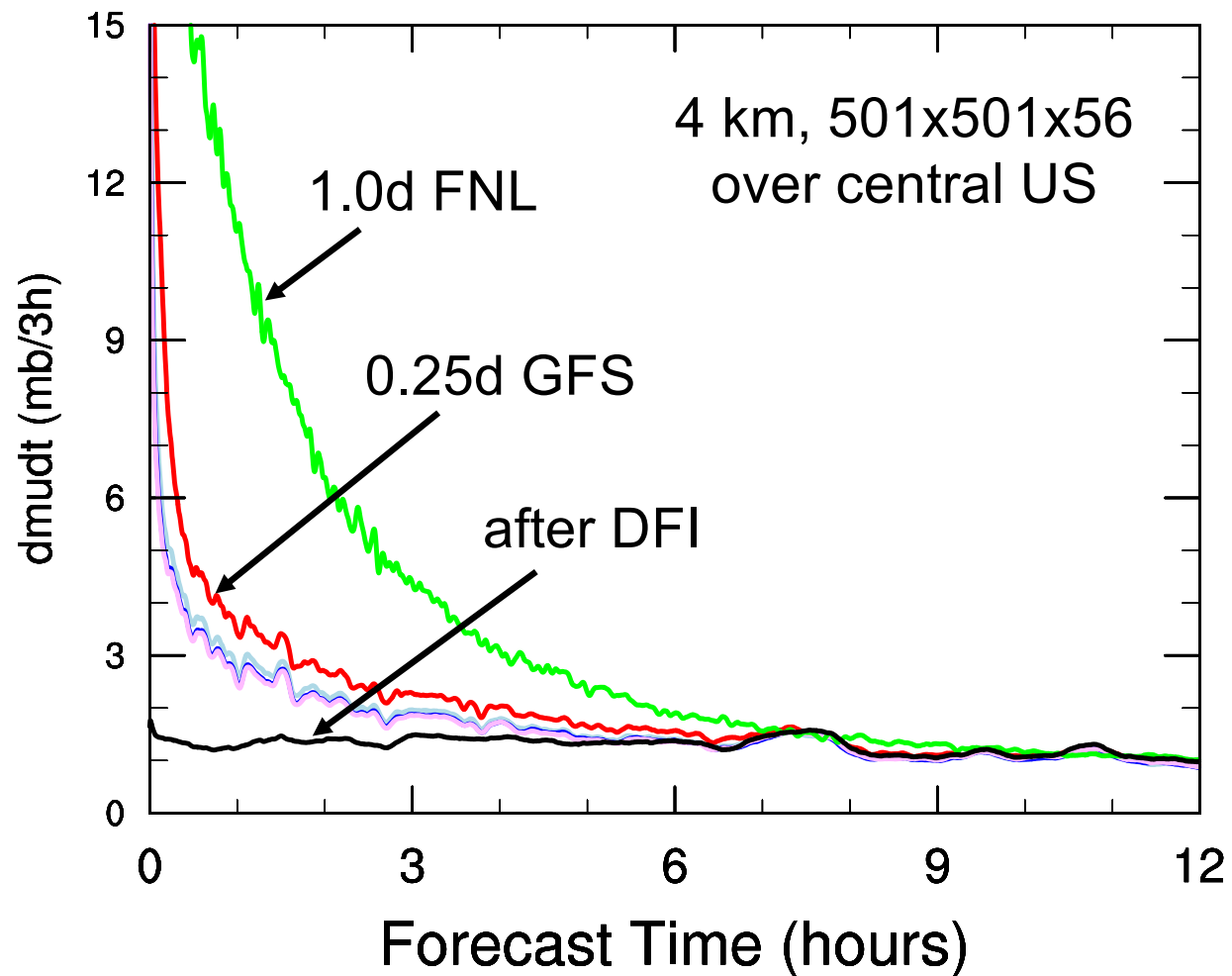
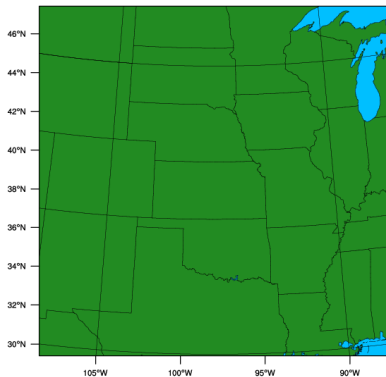
See ‘Run-time Capabilities/**Digital Filter Initialization**’ section of the Users’ Guide.



Digital filter initialization (3)

Use of DFI helps to damp high pressure tendencies in early forecast

4 km WRF domain



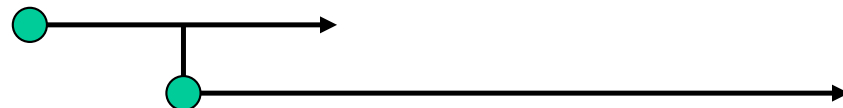
Digital filter initialization (4)

DFL:

(Lynch and Huang, 1994)

$t=0$

Filtering



Forecast

DDFI:

(Huang and Lynch, 1993)

Backward integration



Filtering

Forecast

TDFI:

(Lynch and Huang, 1994)

Backward integration + filtering



Filtering

Forecast



Digital filter initialization (5)

Namelist control: **&dfi_control**

dfi_opt: dfi options: 0: no DFI; 1: DFL; 2: DDFI; 3: TDFI
(recommended)

dfi_nfilter: filter options 0 - 8, recommended: 7

dfi_cutoff_seconds : cutoff period

dfi_write_filtered_input : whether to write
filtered IC to a file

dfi_bckstop_* : stop time for backward integration

dfi_fwdstop_* : stop time for forward integration

Related namelists: examples.namelist



To get pressure tendency data, set **diag_print=1** or **2**

Stochastic parameterization schemes

These are the ways to stochastically perturb forecasts (**&stoch**)

skebs : = 1, activate SKEBS

nens : = N, an integer that controls the random number stream; a different integer will give a differently perturbed forecast

perturb_bdy : = 1, use SKEB pattern; = 2, use user-provided pattern

sppt : = 1, activate stochastically pert parameterization tendencies

spp : = 1, activate stochastically perturbed parameterization physics

Note: sppt and spp only applicable for a subset of physics

Also see ‘Run-time Capabilities/**Stochastic Parameterization schemes**’ section in User’s Guide

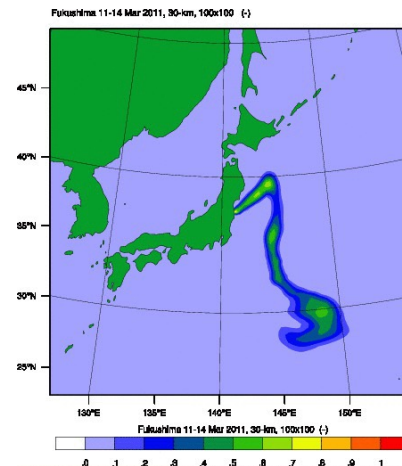
Also see <http://www.cgd.ucar.edu/~berner/skebs.html>



Tracer option

Add the following in **&dynamics** to activate tracer option (default no. is 8: with array names **tr17_1**, **tr17_2**, ..., **tr17_8**):

tracer_opt = 2,



One would need some way to initialize the tracer. A simple initialization can be found in program real (**dyn_em/module_initialize_real.F**)



Trajectory option

Add the following in `&physics` to activate trajectory option:

`traj_opt` = 1,

And set the number of trajectories in `&domains`:

`num_traj` = 1000, (default value)

Since V3.9: it can output meteorological variables, as well as chemistry ones, along the trajectories. Full document at

https://www2.acom.ucar.edu/sites/default/files/wrfchem/Trajectory.desc_.pdf



Additional Output Option

- **wrfout** files: default, data goes to *stream* 0
- Additional output goes to auxiliary output streams
- WRF has 24 *streams*
- Require some knowledge of *registry*
- To output to any of the auxiliary streams, these output specifications need to be added to the namelist under *&domains*

`auxhistN_outname, io_form_auxhistN,
auxhistN_interval, frames_per_auxhistN`



Here **N** is the stream number

Additional Output Option (1)

`prec_acc_dt = 60.:` in `&physics,` unit: minute

Output precipitation in a time interval (e.g. 60 min):

`PREC_ACC_C,` for convective rain

`PREC_ACC_NC,` for explicit rain

`SNOW_ACC_NC,` for explicit snow

Data goes to history file. Interval should be the same as history intervals.



Additional Output Option (2a)

```
&diags  
  p_lev_diag = 1.  
  num_press_levels = 4,  
  press_levels = 85000,70000,50000,20000
```

Output a few met fields on pressure levels :

U_PL, V_PL, S_PL, T_PL, Q_PL, RH_PL, GHT_PL,

Output goes to auxiliary stream 23, so need to set

```
auxhist23_outname, io_form_auxhist23,  
auxhist23_interval, frames_per_auxhist23
```



Additional Output Option (2b)

```
&diags  
  z_lev_diag = 1.  
  num_z_levels = 4,  
  z_levels = 80,150,300,3000
```

Output a few met fields on pressure levels :

U_ZL, V_ZL, S_ZL, T_ZL, Q_ZL, RH_ZL, GHT_ZL,

Output goes to auxiliary stream 22, so need to set

auxhist22_outname, io_form_auxhist22,
auxhist22_interval, frames_per_auxhist22



Additional Output Option (3)

```
&diags  
diag_nwp2 = 1
```

Output a few met fields on model levels :

```
sealevelp, temperature, geoheight,  
pressure, umet, vmet, speed, dir, U10,  
V10, Q2, T2, RAIN, LIQRAIN, TPW, RH
```

Output goes to auxiliary stream 1, so need to set

```
auxhist1_outname, io_form_auxhist1,  
auxhist1_interval, frames_per_auxhist1
```



Additional Output Option (4)

`nwp_diagnostics = 1: (&time_control)`

Output max 10 m wind speed, max helicity in 2 – 5 km layer, max w in updraft and downdraft below 400 mb, mean w in 2 – 5 km layer, and max column graupel in a time window between history output times.

Data goes to history file.



Additional Output Option (5)

```
do_radar_ref = 1: (&physics)
```

Compute and output radar reflectivity using parameters in respective microphysics. Works for options mp_physics = 2,4,6,7,8,10,14,16. Option 9, NSSL mp also produce radar reflectivity output.

Data (**refl_10cm**) goes to history file.



Additional Output Option (6)

`afwa*_opt = 1:` (&afwa, with sub-options)

Output over 60 diagnostic variables to history file (for example, MSLP, precipitable water, cloud cover, visibility, etc.)

See **Registry/registry.afwa** for full listing.

Data goes to history as well as **auxhist2** file. Set `auxhist2_outname`, `io_form_auxhist2`, `auxhist2_interval`, `frames_per_auxhist2`



Additional Output Option (7)

`output_diagnostics = 1: (&time_control)`

Output max, min, time of max and min, mean value, standard deviation of the mean for 8 surface variables (T2, Q2, TSK, U10, V10, 10 m wind speed, RAINCV, and RAINNCV [time step rain]) in history output interval

Data goes to auxiliary stream 3 or **auxhist3**. Set `auxhist3_outname`, `io_form_auxhist3`, `auxhist3_interval`, `frames_per_auxhist3`



Additional Output Option (8)

More climate output (from RASM, `&time_control`):

`mean_diag = 1`: (with interval options)

`diurnal_diag = 1`

Output time-step and diurnal averaging of a number of surface variables and radiative fluxes at surface and top of atmosphere (e.g. monthly averages)

See `run/README.rasm_diag` for details, and `Registry/registry.rasm_diag` for full listing.



Data goes to auxhist5 and auxhist6 files.

Additional Output Option (9)

```
acc_phy_tend = 1: (&physics)
```

Output accumulated physics tendencies for u, v, T, qv, etc.
at history output interval:

**ATHMPTEN, AQVMPTEN, ATHCUTEN, AQVCUTEN, AUC
UTEN, AVCUTEN, ATHBLTEN, AQVBLTEN, AUBLTEN, A
VBLTEN, ATHRATENLW, ATHRATENS...**

- Useful for model diagnostics.
- Data goes to standard history file.



Additional Output Option (10)

`do_avgflx_em = 1: (&dynamics)`

Output history-time-averaged, mass-coupled advective velocities u, v and w:

`AVGFLX_RUM, AVGFLX_RVM, AVGFLX_RWM`

Useful for driving downstream transport model.

Data goes to standard history file.



Time Series Output (1)

- It is a special output in text format with file name like *prefix.d<domain>.TS*
- It outputs 14 surface variables at every time step:
e.g. 10 m u/v, 2 m T/qv, precipitation, radiation fluxes,
surface fluxes
- One file per location (e.g. at weather station), per domain



Time Series Output (2)

- It also outputs profiles of U, V, Th, Qv, PH (levels set by `max_ts_level`, default 15, from surface upward):

prefix.d<domain>.UU

prefix.d<domain>.VV

prefix.d<domain>.TH

prefix.d<domain>.QV

prefix.d<domain>.PH

- One file per variable, per location (e.g. at weather station), and per domain.



Time Series Output (3)

- Requires a file called **'tslist'** present in working directory (a sample of the file is available in `WRF/run/`)
- This file provides a list of names and locations where you would like to output time series. A sample file looks like this:

```
#-----#
# 24 characters for name | pref | LAT | LON |
#-----#
Cape Hallett             hallt -72.330  170.250
McMurdo Station          mcm    -77.851  166.713
```

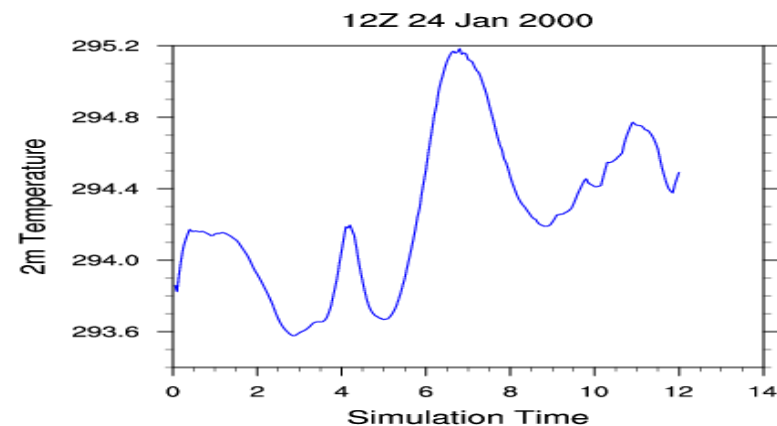
- If output more than 5 locations, use namelist `max_ts_locs` in `&domains`
- More information in `run/README.tslist` and 'WRF Output/Output Diagnostics/**Time Series output**' in UG



Time Series Output (4)

Sample data in **hallt.d01.TS**:

```
Cape Hallett      1  1 hallt ( 36.710, -79.000) ( 41, 38)
(36.600, -79.142) 159.6 meters
  1      0.050000      1  41  38      275.47397      0.00288
  3.52110      -2.34275      99988.76563      244.81276      0.00000
 -29.94841      4.09765      273.90295      278.20197      0.00000
  0.00000      0.00000      1      0.100000      1  41  38
 275.56287      0.00282      3.14414      -2.05875      99956.98438
 244.81276      0.00000      -25.64095      4.18446      273.78323
 278.18314      0.00000      0.00000      0.00000
```



IO quilting: &namelist_quilt

I/O quilting control:

nio_tasks_per_group (>0) : allow IO to be done on separate processors. Performance improvement for large domain runs. A value of 2 to 4 works well.

io_groups (>1) : number of I/O streams that the quilting applies.

See ‘Using IO Quilting’ section, Chap 5, UG

Other ways to improve IO: 1) p-netCDF; 2) use netCDF4 compression option; 3) use io_form_history=102 to output patches of data



Recommended

Start with the **namelist template** in a particular test directory, and the options specified in the file, and make modifications.

WRF User's Guide, examples for various applications and physics suites.

For special applications in ARW, look for related namelists in the file **examples.namelist** in **test/em_real/** directory.

To find which namelist record a namelist variable belongs, try **Registry/Registry.EM_COMMON** and other registry files.

For more information on DFI, adaptive time step and stochastic parameterizations, read WRF Tech Note and User's Guide.

