WRF:
More Runtime Options

Wei Wang

NCAS/NCAR Tutorial, Lincoln UK

October 2019

This material is based upon work supported by the National Center for Atmospheric Research, which is a major facility sponsored by the National Science Foundation under Cooperative Agreement No. 1852977.

More options

- Some useful *runtime* options:
  - Vertical interpolation options (program *real.exe,* &domains)
  - Base state parameters (*real.exe,* &dynamics)
  - Options to use hybrid vertical coordinate (*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)
  - Global runs (&dynamics)
  - Moving nest (&domains)
  - Tracer (&dynamics) / trajectory (&physics, &domains)
  - Stochastic parameterization schemes (&stoch)
  - 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:

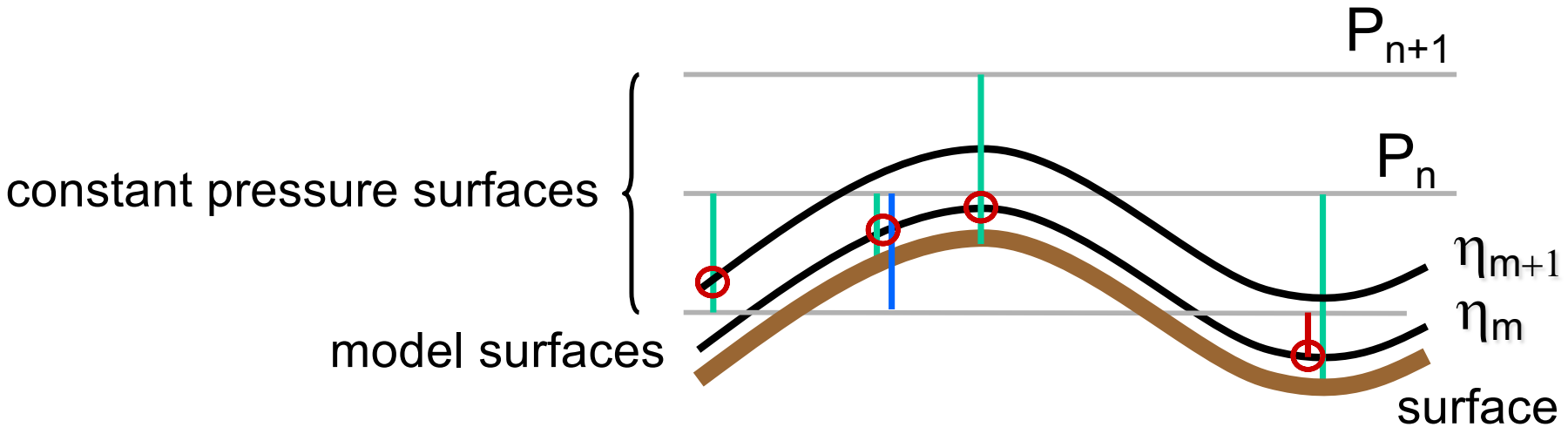
- **&dfi_control**
- **&fdda**
- **&stoch**
- **&diags**
- **&scm**
- **&tc**
- **&noah_mp**

Look for these in **examples.namelist**

Vertical interpolation options (1)

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

- **`interp_type:`** in pressure or log pressure
- **`lagrange_order:`** linear or quadratic
- **`use_surface:`** whether to use surface level data



Vertical interpolation options (2)

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

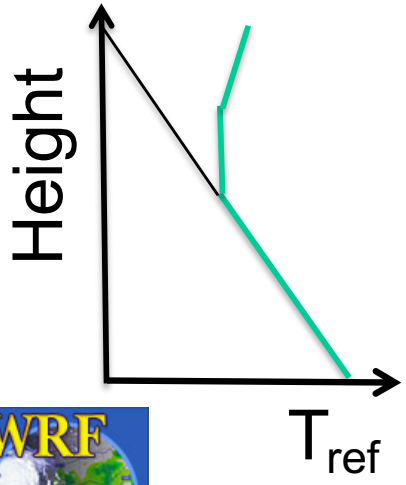
- **`use_levels_below_ground`**: whether to use data below the ground
- **`lowest_lev_from_sfc`**: logical, whether surface data is used to fill the lowest model level values
- **`force_sfc_in_vinterp`**: number of levels to use surface data, default is 1
- **`extrap_type`**: how to do extrapolation: 1 - use 2 lowest levels; 2 - constant
- **`t_extrap_type`** : extrapolation option for temperature: 1 - isothermal; 2 - 6.5 K/km; 3 - adiabatic

Look for these in **`examples.namelist`**

Base State Parameters

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

`base_temp`	Base state surface temperature
`iso_temp`	Base state stratosphere temperature (default 200 K)
`base_pres_strat`	Pressure at which the stratosphere temperature lapse rate changes (since 3.6.1)



Help to improve simulations when model top is higher than 20 km (~ 50 mb)

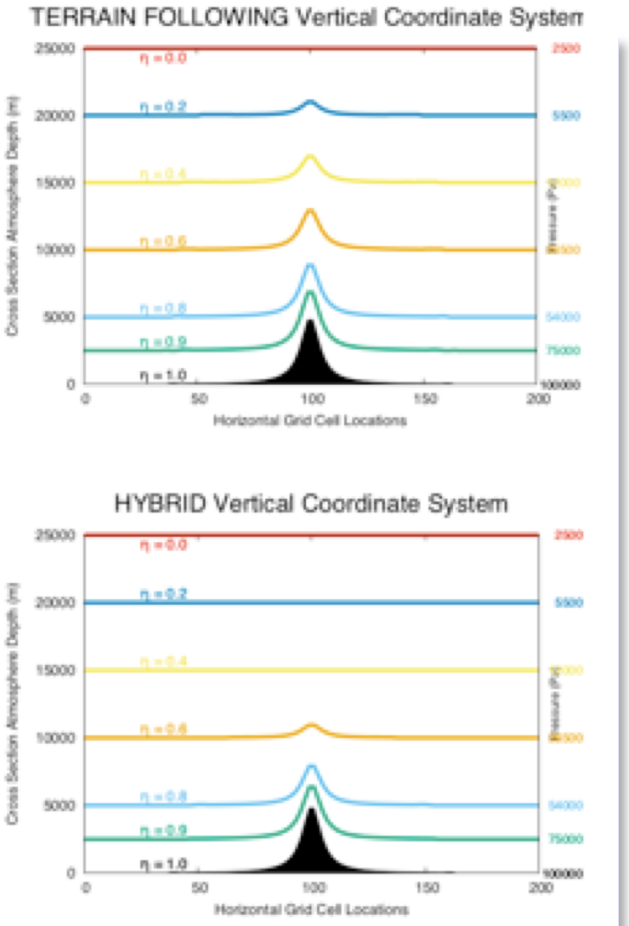
Hybrid Vertical Coordinate Option

- Decision made when running program **`real.exe`**, by setting these namelists in &dynamics

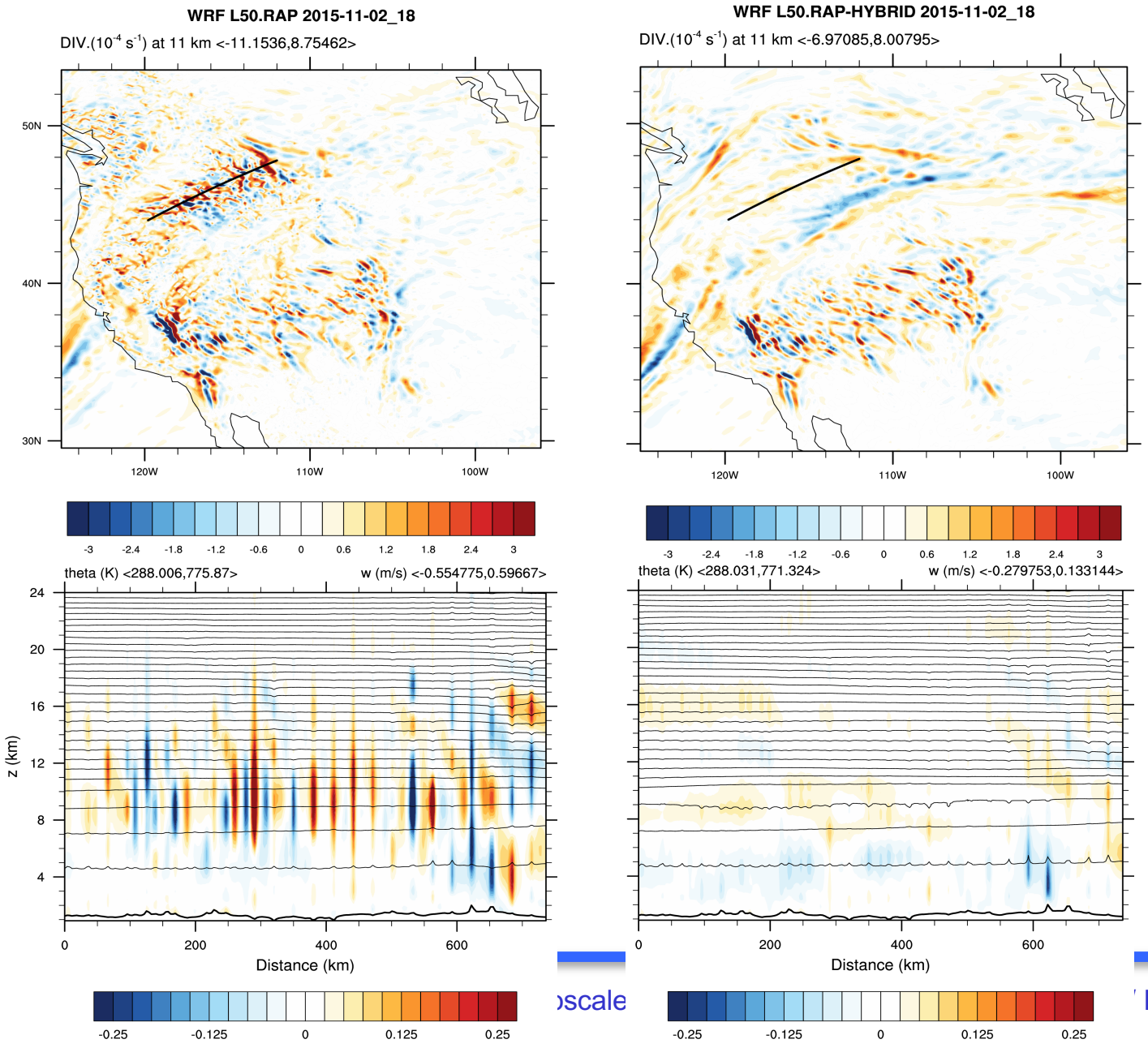
  `hybrid_opt = 2` (0 turns it off)

  `eta_c = 0.2` (default)

- New since V3.9
- Default in V4.0

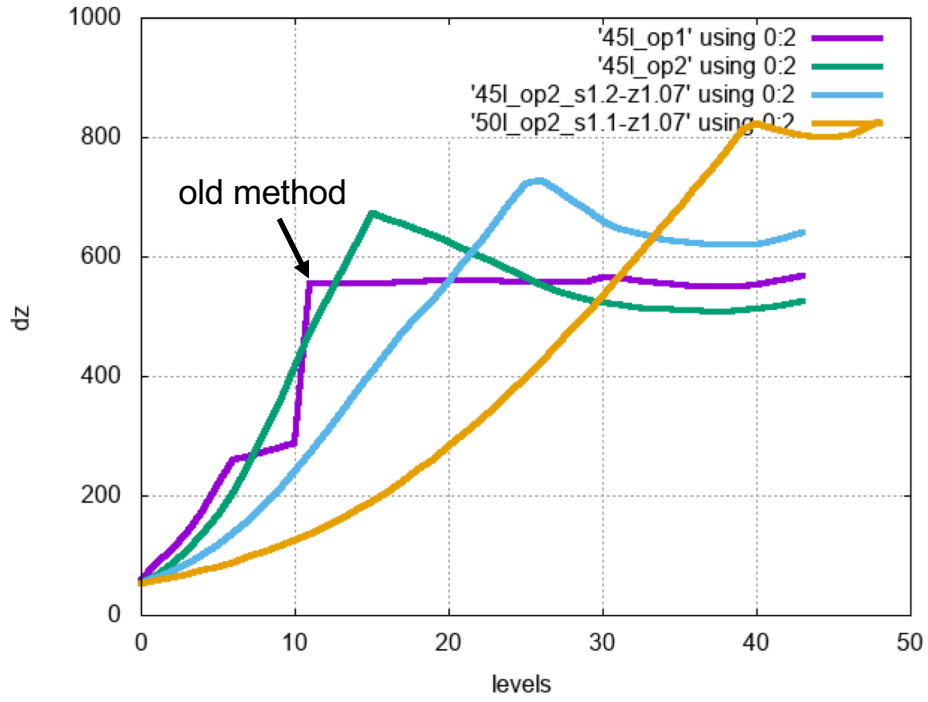


Hybrid Vertical Coordinate Options



Defining Vertical Levels

```
&domains
auto_levels_opt = 1 or 2
dzstretch_s = 1.1 ~ 1.3
dzstretch_z = 1.02 ~ 1.1
dzbot = 20 ~ 50
max_dz = < 1000
```



IO Control (1)

History output control in **`&time_control`**

`history_interval`:	used often, unit in minutes
`history_interval_h`:	history output interval in hours
`history_interval_s`:	history output interval in seconds
`history_begin_h`:	history output beginning time in hours
`history_begin_d`:	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 = "/mydata/met_em.d<domain>.<date>"`
: explicitly specify input file (it name and directory)

`history_outname = "/mydata/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 history output in **`&time_control`**

1. Change Registry.EM 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
```

IO Control (4)

Starting in V3.2, there is an alternative to add/remove output fields at **runtime** (state variables in Registry only)

1. new 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
```

See 'Run-Time IO' section in Chapter 5, 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,
```

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>"

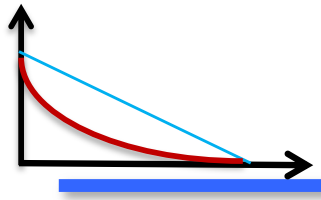
`auxinput4_interval` = 360, 360,

`io_form_auxinput4` = 2 (netCDF)

See 'Using sst_update Option' in Chapter 5, 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

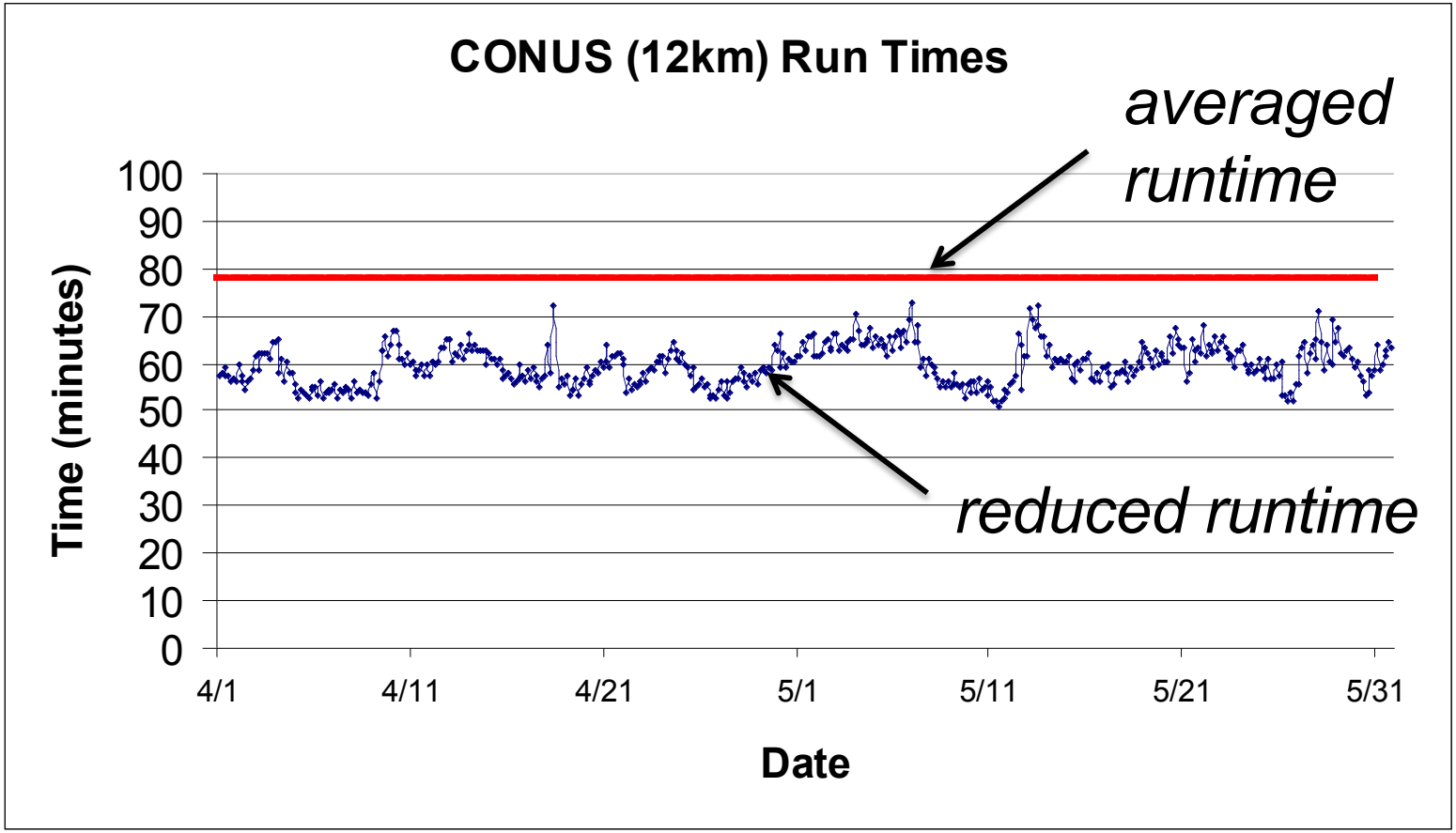


Adaptive time steps (1)

- 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 ‘Using Adaptive Time Stepping’ section in Chapter 5, UG

Adaptive time steps (2): an example



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

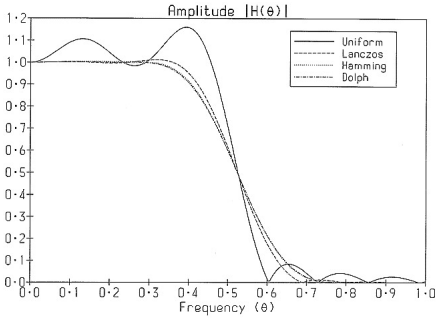
Adaptive time steps (3)

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

`use_adaptive_time_step`	logical switch
`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*DX
`max_time_step`	in seconds; e.g. set to 8*DX
`min_time_step`	in seconds; e.g. set to 4*DX

Digital Filter Initialization (DFI) (1)

- DFI is a way to use a low-pass filter to improve model initial conditions
- Useful for short-range model runs (1-6 hours)
- 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 precipitation, numerical instability and degrade subsequent data assimilation



Digital filter initialization (2)

Using DFI

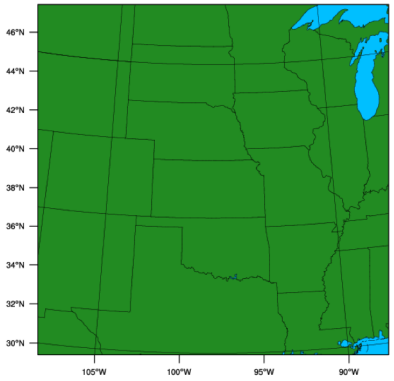
- can construct consistent model fields which do not exist in the initial conditions, e.g. vertical motion, cloud variables
- may reduce the spin-up problem in early simulation hours
- Useful for short-range (1-6 h) forecasts and cycling with data assimilation

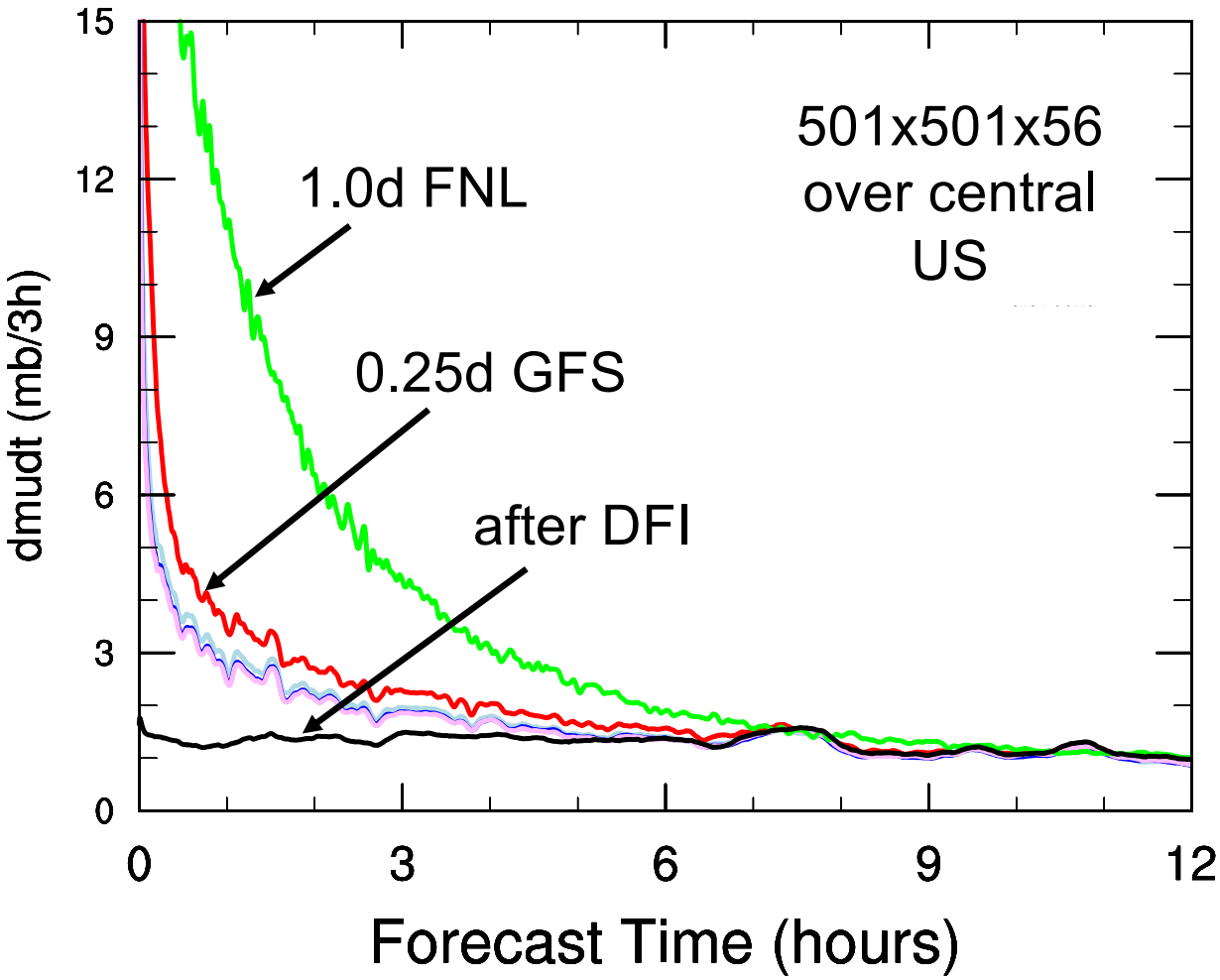
DFI is done after program **`real`**, or data-assimilation step

See 'Using Digital Filter Initialization', Chap 5, UG.

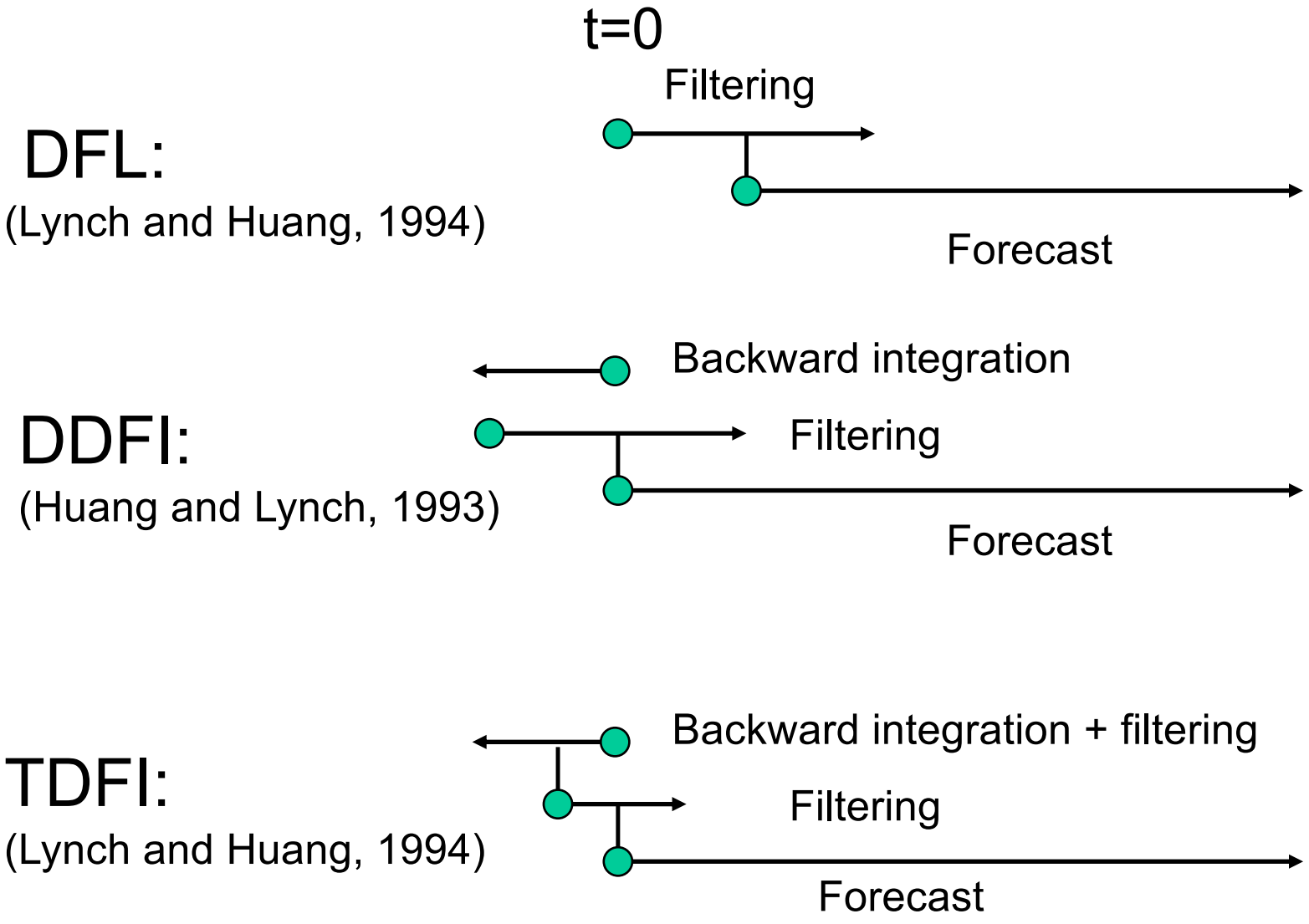
Digital filter initialization (3)

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





Digital filter initialization (4)



Digital filter inilialization (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
- **`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`**

Global application

- Setup in WPS:

  `map_proj = 'lat-lon'`

  `e_we, e_sn:` **`geogrid`** will compute dx, dy

  See template `'namelist.wps.global'`

- Requires only one-time period data
- In the model stage: (&dynamics)

  `fft_filter_lat:` default value is 45 degrees

  Caution: some options do not work, or have not been tested with global domain. Start with template `'namelist.input.global'`

See 'Global Run' section, Chap 5, UG

Automatic moving nest options

Tropical cyclone / typhoon / hurricane applications: (&domains)

- **`vortex_interval:`** time interval when vortex location is estimated
- **`max_vortex_speed:`** used to compute the search radius for vortex location
- **`corral_dist:`** how far the vortex can move near the parent domain boundary (number of grids)
- **`track_level:`** e.g. 700 or 500 mb
- **`time_to_move:`** hold nests still until this time

See ‘Moving Nested Run’, Chap 5, UG

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)

New in V3.9: it can output meteorological variables, as well as chemistry ones, along the trajectories.

Stochastic parameterization schemes

This is a way to stochastically perturb forecasts (&stoch)

- **skebs:** = 1, activate the scheme
- **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 (new in 3.5)
- **sppt:** = 1, activate stochastically parameterized pert tendencies
- **spp:** = 1, activate stochastic perturbed parameters in physics

Also see 'Option to stochastically perturb forecasts' section in Chap 5, UG

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

Additional Output Option (1)

`prec_acc_dt = 60.:` in **`&physics`**

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

(Caution: *May not suitable for use in long runs*)

Additional Output Option (2a)

Since V3.4.1:

```
&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)

Since V3.7.1:

```
&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 23, so need to set

auxhist23_outname, io_form_auxhist23,

auxhist23_interval, frames_per_auxhist23

Additional Output Option (3)

Since V3.9:

```
&diags
 diags_nwp = 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)

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])

```
auxhist3_outname =“wrfxtrm_d<domain>_<date>”
io_form_auxhist3 = 2
auxhist3_interval = 1440, 1440,
frame_per_auxhist3 = 10, 10,
```

Additional Output Option (5)

`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 (6)

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

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

Data goes to history file.

Additional Output Option (7)

`do_avgflx_em = 1:` **`(&dynamics)`**

output history-time-averaged, column-pressure-coupled u, v and w:
AVGFLX_RUM, AVGFLX_RVM, AVGFLX_RWM
– useful for driving downstream transport model

Additional Output Option (8)

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

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

See Registry/registry.afwa for full listing.

Data goes to history as well as auxhist2 file.

Additional Output Option (9)

More climate output (from RASM, new in V3.9):

`mean_diag = 1:` (with interval options, &time_control)

`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.

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

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):

  `prefix.d<domain>.UU`

  `prefix.d<domain>.VV`

  `prefix.d<domain>.TH`

  `prefix.d<domain>.QV`

  `prefix.d<domain>.PH`

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

Time Series Output (3)

- Not a namelist option to turn it on
- If output more than 5 locations, use namelist **`max_ts_locs`** in **`&domains`**
- Requires a file called '**`tslist`**' present in working directory (a sample of the file is available in **`WRF/run/`**

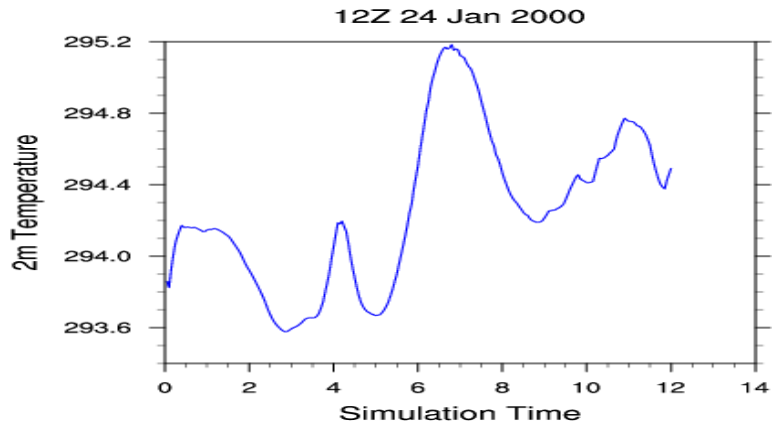
```
#-----------------------------------------#
# 24 characters for name | pfx |  LAT   |   LON   |
#-----------------------------------------#
Cape Hallett             hallt -72.330  170.250
McMurdo Station          mcm   -77.851  166.713
```

- This file provides a list of locations where you would like to output time series
- More information in **`run/README.tslist`** and 'Output Time Series' section, Chapter 5, UG

Time Series Output (4)

Content 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
```



Recommended

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

Chapter 5 of ARW User's Guide, pages 5-37 – 5-39: examples for various applications; page 34: physics suites.

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

For more information on global extension, DFI and adaptive time step, read Tech Note, and User's Guide.