

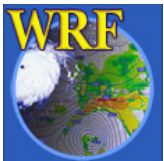


WRF: *More Runtime Options*

Wei Wang

NCAR/NESL/MMM

July 2013



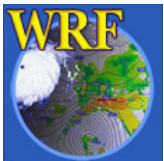
Fortran namelist

- Fortran 90 namelist is used to configure a run without recompiling the code
- Fortran 90 namelist has very specific format, so edit with care:

`&namelist-record` - start

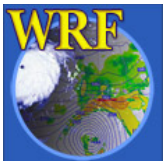
/ - end

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



More options

- Some useful runtime options:
 - IO options
 - Vertical interpolation options
 - Options for long simulations
 - Adaptive-time step
 - Digital filter
 - Global runs
 - Moving nest
 - TC options
 - Tracer / trajectory
 - Optional output
 - Stochastic kinetic-energy backscatter scheme (SKEB)
 - IO quilting
- Time series output (surface and profile)



namelist.input

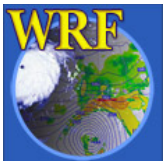
general namelist
records:

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

specialized namelist
records:

`&dfi_control`
`&fdda`
`&grib2`
`&scm`
`&tc`
`&noah_mp`

Look for these in
`examples.namelist`



IO Control (1)

History output control in `&time_control`

`history_interval`: used often, unit in minutes

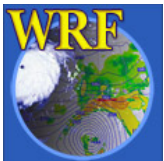
`history_interval_h`: history output interval in hours

`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

Complete listing in `Registry/`
`registry.io_boilerplate`



IO Control (2)

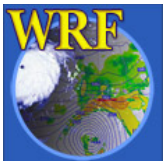
Optional history output in `&time_control`

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 (3)

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

1. new namelists in &time_control:

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

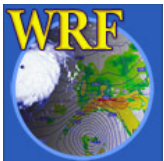
2. prepare a text file ('my_output.txt') to select io fields:

```
+: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



Vertical interpolation options (1)

Program **real** for ARW only, optional, &domains:

use_surface: whether to use surface observations

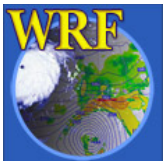
use_levels_below_ground: whether to use data below the ground

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



Vertical interpolation options (2)

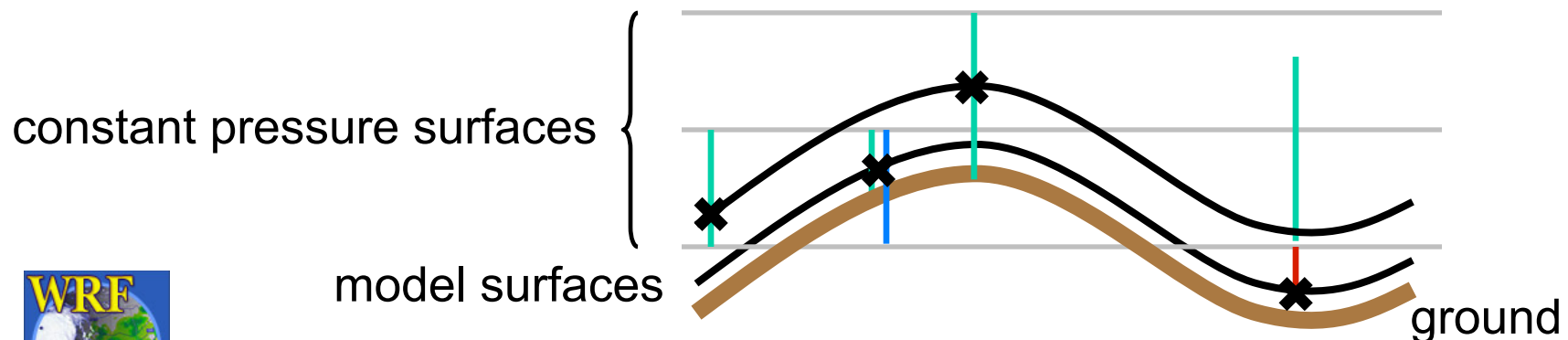
Program **real** for ARW only, optional:

interp_type: in pressure or log pressure

lagrange_order: linear or quadratic

zap_close_levels: Δp where a non-surface pressure level is removed in vertical interpolation

related namelists: **examples.namelist**



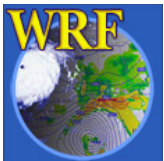
Vertical interpolation options (3)

Since V3.3.1:

interp_theta Whether to interpolate T or theta;
= .false., interpolate T, default

hypsonetric_opt An alternative way to obtain
geopotential height in program real,
and pressure in the model.
= 2, default since V3.4

→ Help to improve biases introduced in
vertical integration



Options for long simulations (1)

Lower boundary update control: allow SST, seaice, monthly vegetation fraction, and albedo to be updated regularly during a model run:

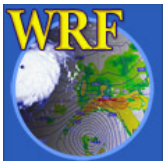
sst_update: 0 – no update
1 – update all above fields

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

Other namelists required in **&time_control**:

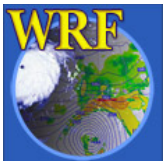
auxinput4_inname = “wrflowinp_d<domain>”
auxinput4_interval = 360, 360,
io_form_auxinput4 = 2 (netCDF)

See ‘**Using sst_update Option**’ in Chapter 5, User’s Guide



Options for long simulations (2)

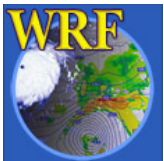
<code>sst_skin</code>	diurnal water temp update
<code>tmn_update</code>	deep soil temp update, used with <code>lagday</code>
<code>lagday</code>	averaging time in days
<code>bucket_mm</code>	bucket reset value for rainfall
<code>bucket_j</code>	bucket reset value for radiation fluxes
<code>spec_exp</code>	exponential multiplier for boundary zone ramping (set in <i>real</i>). Usually used with wider boundary zone



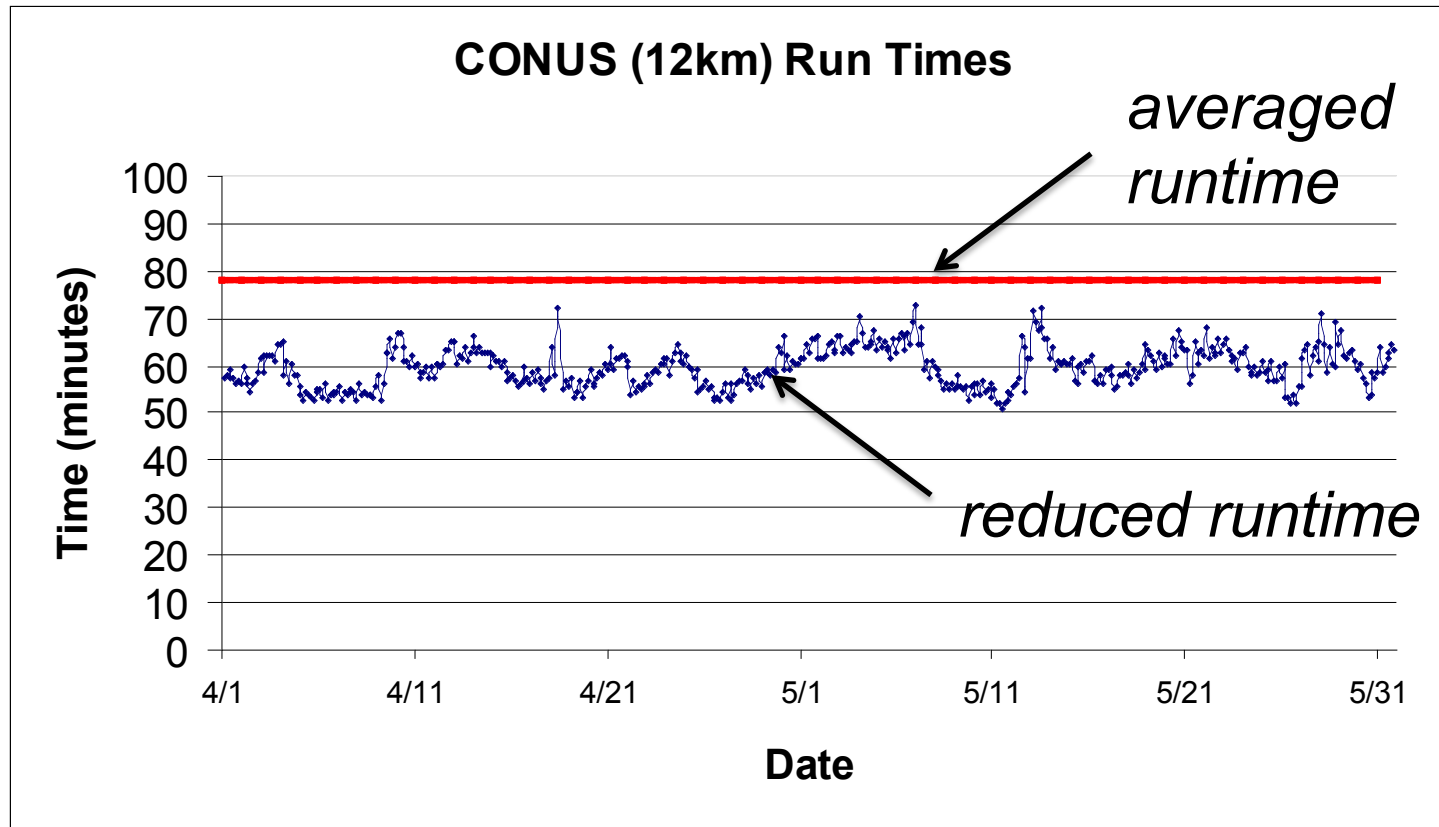
Adaptive time steps (1)

- Adaptive-time-step is a way to maximize the model time step while keeping the model numerically stable.
- New in V3. Good to use for real-time run.
- May not work in combination with other options.

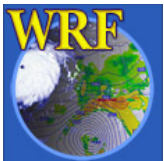
Also see ‘**Using Adaptive Time Stepping**’
section in Chapter 5, UG



Adaptive time steps (2): an example



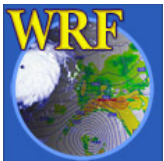
On average, forecasts finish in 60 min (50-73min)
as compared to 79 min standard runtime



Adaptive time steps (3)

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

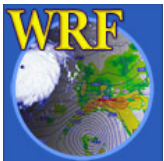
<code>use_adaptive_time_step</code>	logical switch
<code>step_to_output_time</code>	whether to write at exact history output times
<code>target_cfl</code>	maximum cfl allowed (1.2)
<code>max_step_increase_pct</code>	percentage of time step increase each time; set to 5, 51, 51 (larger value for nest)
<code>starting_time_step</code>	in seconds; e.g. set to 4*DX
<code>max_time_step</code>	in seconds; e.g. set to 8*DX
<code>min_time_step</code>	in seconds; e.g. set to 4*DX



Digital filter initialization (1)

Digital filter initialization is a simple way to remove initial model imbalance:

- May be introduced by simple interpolation, different topography, or by objective analysis, or data assimilation
- It may generate spurious gravity waves in the early simulation hours, which could cause erroneous precipitation, numerical instability and degrade subsequent data assimilation

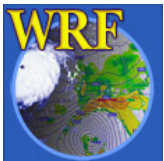


Digital filter initialization (2)

Using DFI

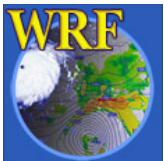
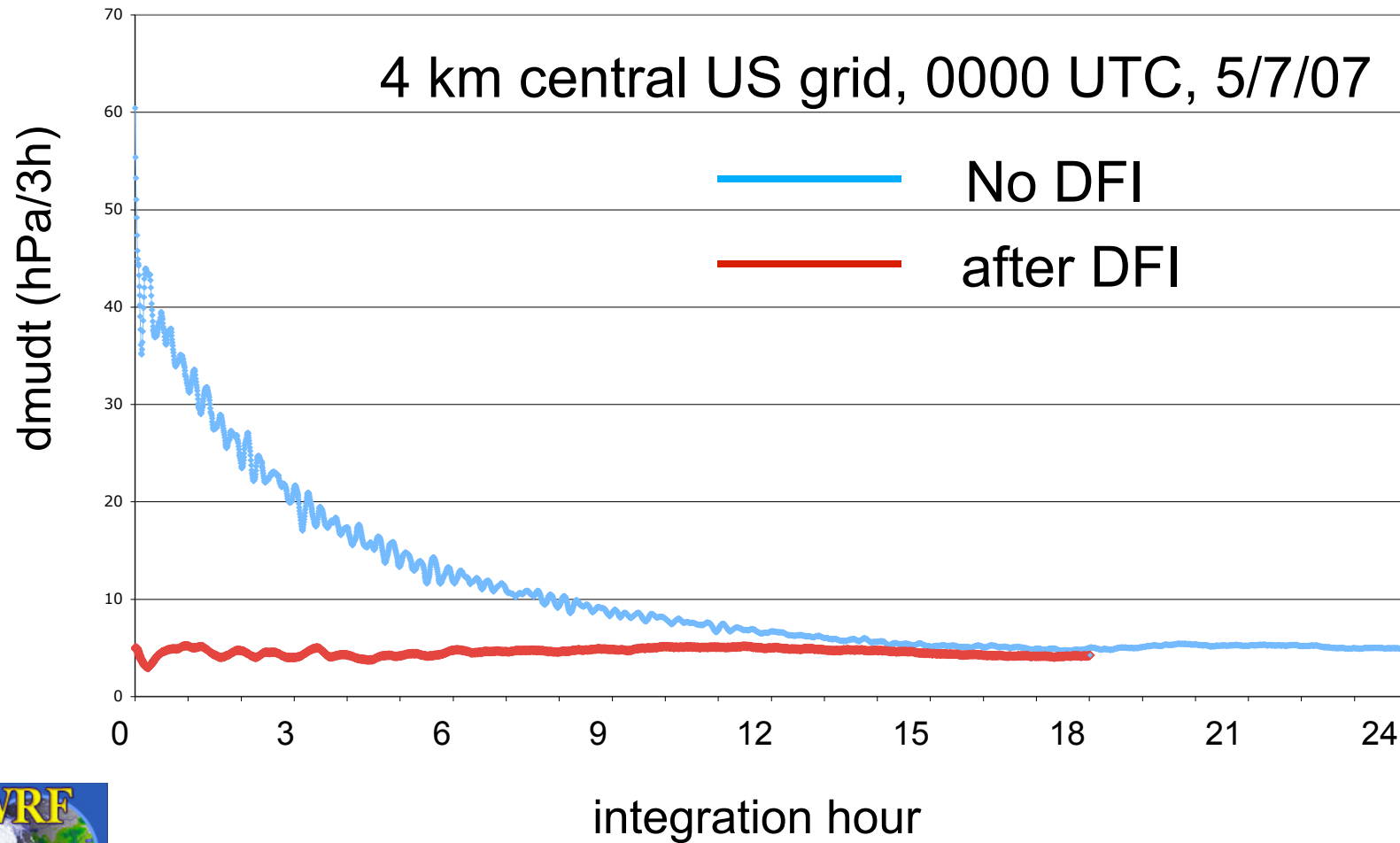
- can construct consistent model fields which do not exist in the initial conditions, e.g. vertical motion, cloud variables
- may reduce the spin-up problem in early simulation hours
- Useful if the first few hours of forecast is the interest

DFI is done after program **real**, or data-assimilation step, just before model integration



See ‘**Using Digital Filter Initialization**’, Chap 5, UG.

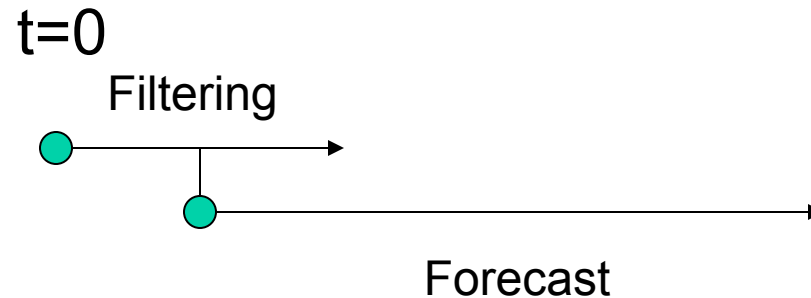
Digital filter initialization (3)



Digital filter initialization (4)

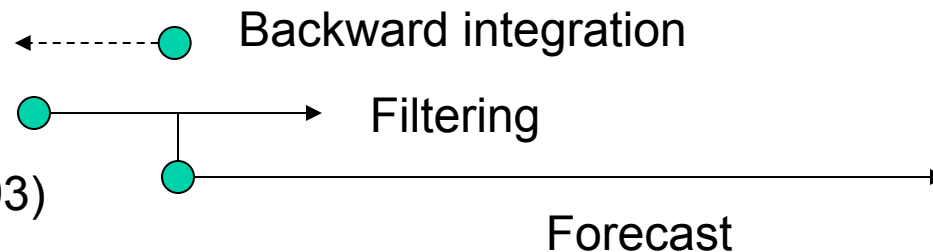
DFL:

(Lynch and Huang, 1994)



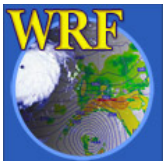
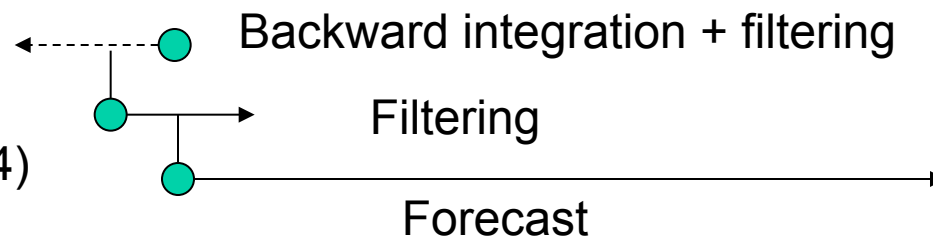
DDFI:

(Huang and Lynch, 1993)



TDFI:

(Lynch and Huang, 1994)



Digital filter initialization (5)

Namelist control: **&dfi**

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

dfi_nfilter: filter options 0 - 8, recommended: 7

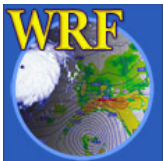
dfi_cutoff_seconds : cutoff period

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**



Global application

Setup mostly done in WPS:

`map_proj = 'lat-lon'`

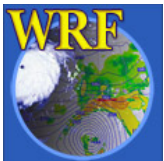
`e_we, e_sn`: `geogrid` will compute `dx, dy`

See template `'namelist.wps.global'`

In the model stage:

`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:

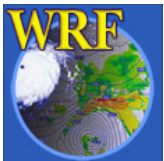
vortex_interval: time interval when vortex location is estimated

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

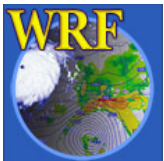
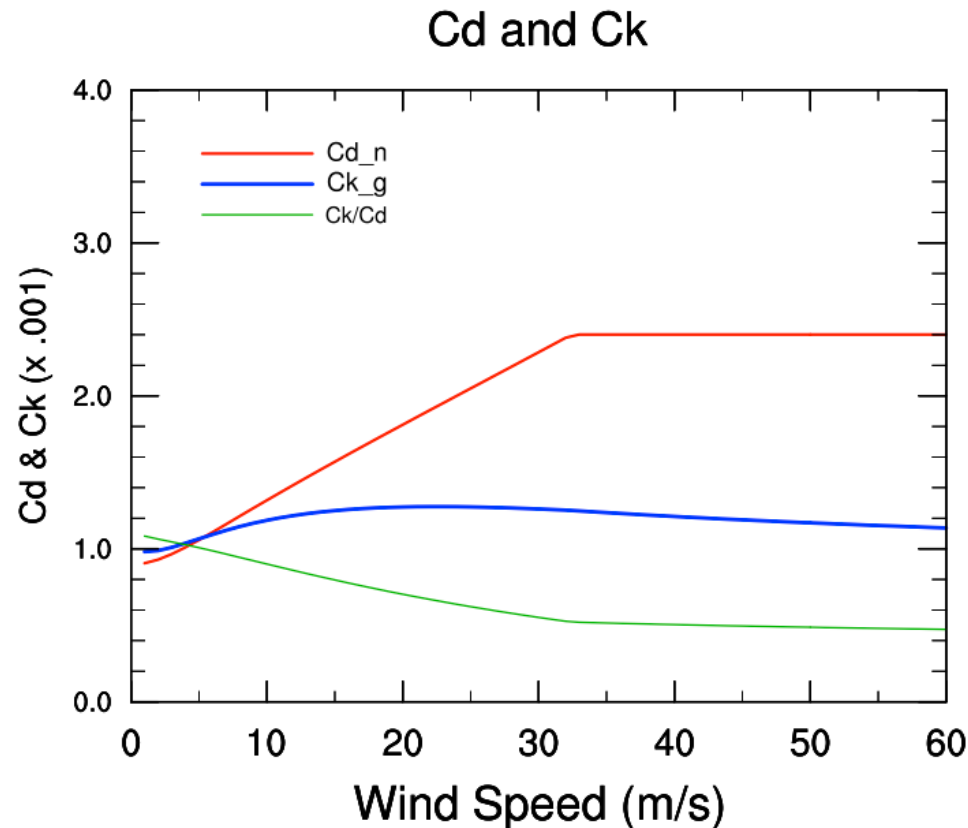
time_to_move: hold nests still until this time



See ‘**Moving Nested Run**’, Chap 5, UG

TC options (1)

isftcflx: alternative C_d (Donelan) and C_k ($=2$, Garratt) formulation for TC application



TC options (2)

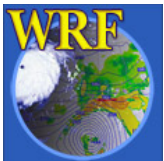
sf_ocean_physics: simple ocean mixed layer

oml_hml0: initial ocean mixed layer depth

oml_gamma: lapse rate in deep water

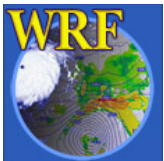
The ocean mixed layer model can also be initialized with real-data, e.g. HYCOM. More info can be found at

http://www.mmm.ucar.edu/wrf/users/hurricanes/wrf_ahw.html



TC options (3)

sf_ocean_physics = 2: 3D Price-Weller-Pinkel (PWP) ocean model based on Price et al. (1994). It has full ocean process (e.g. advection, pressure-gradient force, and mixing). It doesn't have ocean topography. Only simple initialization is provided in Version 3.5.

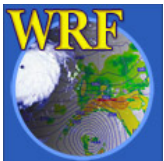


tracer option

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.



trajectory option

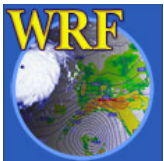
Add the following in `&physics` to activate trajectory option:

```
traj_opt = 1,
```

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

```
num_traj = 25, (default value)
```

Output: `traj_i(num_traj)`, `traj_j`, `traj_k`,
`traj_lat`, `traj_long`



Stochastic kinetic-energy backscatter scheme

This is a way to stochastically perturb forecasts.

stoch_force_opt: = 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)

Also see ‘**Option to stochastically perturb forecasts**’ section in Chap 5, UG

(Berner et al. 2011, MWR)



Additional Output Option (1)

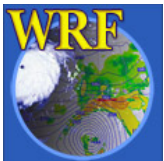
`prec_acc_dt = 60.:`

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



Additional Output Option (2)

Since V3.4.1:

`&diags`

`p_lev_diag = 1.`

`num_press_levels = 4,`

`press_levels = 85000,70000,50000,20000`

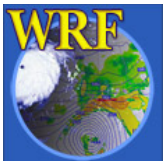
Output a few met fields on pressure levels :

`U_PL, V_PL, S_PL, T_PL, TD_PL, RH_PL, GHT_PL,`

Output goes to auxiliary stream 23, so need to set

`auxhist23_outname, io_form_auxhist23,`

`auxhist23_interval, frame_per_auxhist23`



Additional Output Option (3)

`output_diagnostics = 1:`

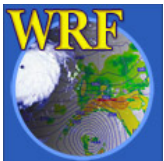
output max, min, time of max and min, mean value, standard deviation of the mean for 8 surface variables (T2, Q2, TSK, U10, V10, 10 m wind speed, RAINCV, and RAINNCV [time step rain])

`auxhist3_outname = "wrfxtrm_d<domain>_<date>"`

`io_form_auxhist3 = 2`

`auxhist3_interval = 60`

`frame_per_auxhist3 = 10`

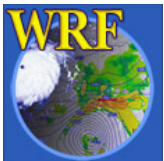


Additional Output Option (4) *(new)*

`nwp_diagnostics = 1:`

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.



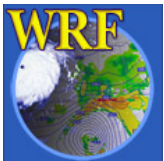
Additional Output Option (5)

`do_avgflx_em = 1:`

output history-time-averaged, column-pressure-coupled u, v and w:

`AVGFLX_RUM, AVGFLX_RVM, AVGFLX_RWM`

– useful for driving downstream transport model



IO quilting: &namelist_quilt

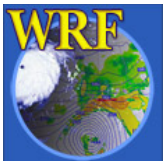
Parallel I/O control:

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.

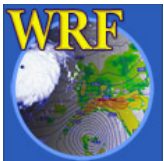
See ‘**Using IO Quilting**’ section, Chap 5, UG

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, surface fluxes
- It also outputs profiles of U, V, Th, Qv, PH
- One file per location (e.g. at weather station)

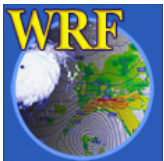


Time Series Output (2)

- Not a namelist option
- If output more than 5 locations, use namelist **max_ts_locs**
- Depends the presence of a file called '**tslist**' (a sample of the file is available in **WRFV3/run/**)

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

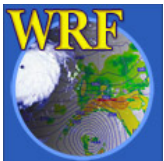
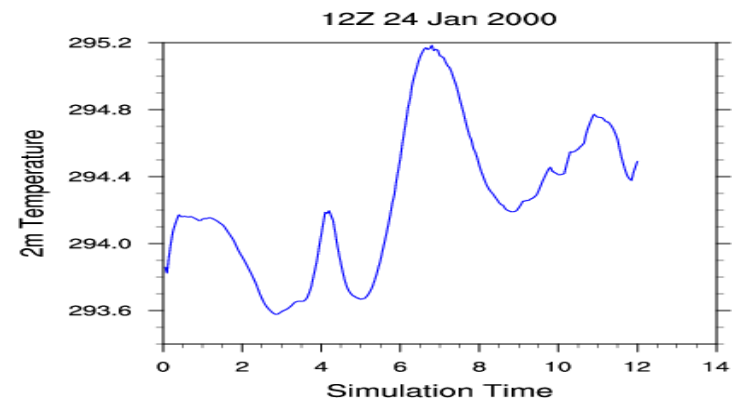
- This file provides a list of locations where you would like to output time series
- More information in **run/README.tslist** and '**Output Time Series**' section, Chapter 5, UG



Time Series Output (3)

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 5-25 – 5-27: examples for various applications.

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.

