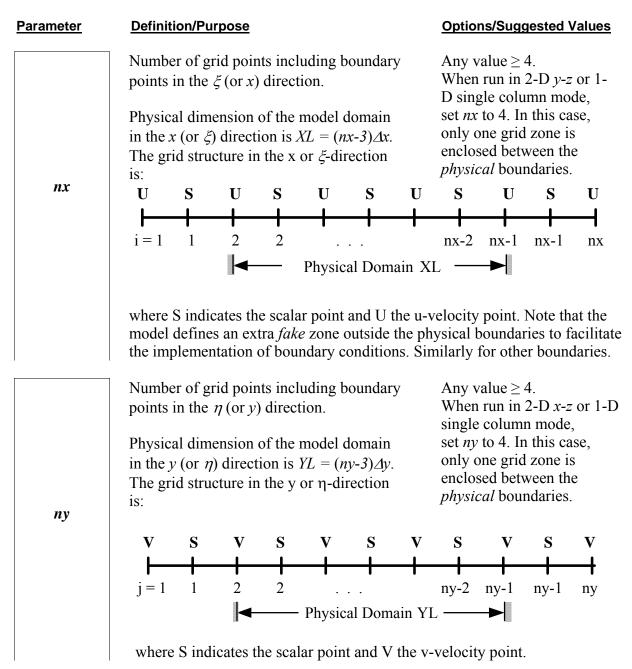# 4 ARPS 5.0 Runtime / Configuration Reference Guide

ARPS provides its users with flexible control over many parameters that can be used to configure the model for various applications. Most of these parameters can be set at run time without modifying or recompiling the model code. These parameters are specified in a model input file, *arps.input*. The input file is in NAMELIST format, and is read in by ARPS via standard input (FORTRAN I/O channel 5). The NAMELIST format offers the flexibility of optional specification of certain parameters. When a parameter is missing, its default value specified inside subroutine INITPARA is used.
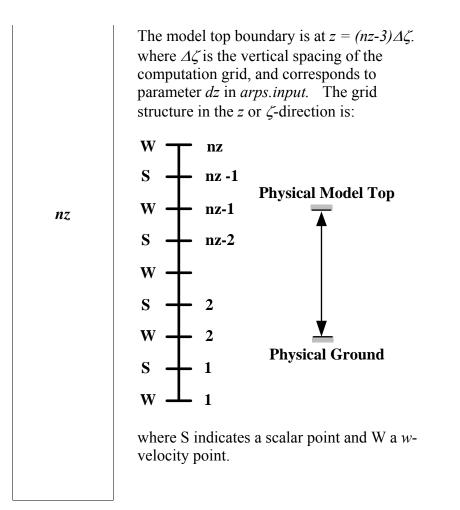
In addition to the parameters used for model execution, input file *arps.input* also contains NAMELIST blocks for parameters that are used only by other programs. In other words,, *arps.input* is shared by a number of programs in the ARPS package, which simplifies input file maintenance and ameliorates the need for specification of the same parameters in multiple files.

This chapter provides a reference guide on how to choose appropriate values for these control parameters. Recommended values are provided for some parameters. The control parameters are organized into NAMELIST blocks in the input files. The names of these blocks are given in the group headers.

Model Dimension Parameters
(&grid_dims)

Parameter	**Definition/Purpose**	**Options/Suggested Values**

nx

Number of grid points including boundary points in the ξ (or x) direction.

Physical dimension of the model domain in the x (or ξ) direction is $XL = (nx\text{-}3)\varDelta x$. The grid structure in the x or ξ-direction is:

U   S   U   S   U   S   U   S   U   S   U

i = 1   1   2   2   . . .   nx-2   nx-1   nx-1   nx

Physical Domain XL

where S indicates the scalar point and U the u-velocity point. Note that the model defines an extra *fake* zone outside the physical boundaries to facilitate the implementation of boundary conditions. Similarly for other boundaries.

Any value ≥ 4. When run in 2-D *y-z* or 1-D single column mode, set *nx* to 4. In this case, only one grid zone is enclosed between the *physical* boundaries.

ny

Number of grid points including boundary points in the η (or y) direction.

Physical dimension of the model domain in the y (or η) direction is $YL = (ny\text{-}3)\varDelta y$. The grid structure in the y or η-direction is:

V   S   V   S   V   S   V   S   V   S   V

j = 1   1   2   2   . . .   ny-2   ny-1   ny-1   ny

Physical Domain YL

where S indicates the scalar point and V the v-velocity point.

Any value ≥ 4. When run in 2-D *x-z* or 1-D single column mode, set *ny* to 4. In this case, only one grid zone is enclosed between the *physical* boundaries.

Number of grid points in the ζ (or z) direction.

Any value ≥ 4.

The model top boundary is at $z = (nz-3)\Delta\zeta$. where $\Delta\zeta$ is the vertical spacing of the computation grid, and corresponds to parameter *dz* in *arps.input*. The grid structure in the *z* or ζ-direction is:

nz

W		**nz**
S		**nz -1**
W		**nz-1**
S		**nz-2**
W		
S		**2**
W		**2**
S		**1**
W		**1**

Physical Model Top

Physical Ground

where S indicates a scalar point and W a *w*-velocity point.

Parameters used by Message Passing (MPI) Run Ignored by non-MPI run of ARPS (&message_passing)

Parameter	Definition/Purpose	Options/Suggested Values
nproc_x *nproc_y*	Number of processors in the *x* and *y* directions, respectively, in the domain-decomposition of the physical domain for an ARPS MPI run. Note that (*nx-3*)/*nproc_x* and (*ny-3*)/*nproc_y* must be integer. *nproc_x* × *nproc_y* is the total number of processors for arps_mpi.	Integer values ≥ 1.

Parameter	Definition/Purpose	Options/Suggested Values
max_fopen	Maximum number of files allowed open when reading or writing. Set *max_fopen* ≥ number of processors if *initopt*=1 (when needing to read in a sounding file). Otherwise, a smaller number than the number of processors can be used to improve I/O efficiency, especially when *nproc_x × nproc_y* is large.	*max_fopen = nproc_x × nproc_y works most of the time.*

Parameters used by Adaptive Grid Refinement version of ARPS Available in ARPS 4.5 but not available in ARPS 5.0 currently (&arpsagr)

Parameter	Definition/Purpose	Options/Suggested Values
	To be added later.	

Comment Strings on Current Run (&comment_lines)

Parameter	Definition/Purpose	Options/Suggested Values
nocmnt *cmnt*	*cmnt* is an array of character strings containing user comments for this run. These comments will accompany the history data dumps. *nocmnt* is the number of comment strings to read in and carried in the data. The maximum array size is 50 and maximum string length is 80. CHARACTER (LEN=80) :: *cmnt(50)*	User specified strings of maximum length of 80.

Model Run Name
(&jobname)

Parameter	**Definition/Purpose**	**Options/Suggested Values**
runname	A character string containing the identifying information for this run. The initial characters before a blank space or a comma will be used to construct file names of a number of output files, including the history and restart data. This string is also contained within the history data file and placed at the bottom of graphics pages produced by ARPSPLT. CHARACTER (LEN=80) :: *runname*.	Use up to 80 characters to identify this experiment and data files to be produced, *e.g.*, *runname='arpstest'*.

Model Geometry Configuration Parameter
(&model_configuration)

Parameter	**Definition/Purpose**	**Options/Suggested Values**
runmod	Option for running ARPS in 1D, 2D or 3D mode. For *runmod* = 2, set *ny*=4 and periodic conditions for the north and south boundary. For *runmod* = 3, set *nx*=4 and periodic conditions for the east and west boundary. For *runmod* = 4, set *nx*=*ny*=4, and periodic conditions for all four lateral boundaries. When *runmod* is 2-D or 1-D (runmod>1), the analytic initial perturbation and analytic mountain are automatically set to 2-D or 1-D.	1 = model run in 3-D. 2 = model run in 2-D *x-z* slice mode. 3 = model run in 2-D *y-z* slice mode. 4 = model run in 1-D single column mode.

Model Initialization Parameters
(&initialization)

Parameter	Definition/Purpose	Options/Suggested Values
initime	A character string specifying the calendar day and time (UTC) corresponding to the model clock time zero. *initime = 'yyyy-mn-dd.hh:mm:ss'*, where *yyyy* , *mn*, *dd*, *hh, mm* and *ss* are integers for year, month, day, hours, minutes and seconds. CHARACTER (LEN=19) :: *initime*	Character string in quotes. Choose to match the model time zero, not necessarily the time when the model is started or restarted. *e.g., initime ='2003-01-01.12:00:00'.*
timeopt	Option to check the consistency of user specified time parameters, *initime* and *tstart*, with the time in the history data used initialize ARPS when *initopt*=3, and adjustment when desired.	= 1, warning on inconsistency and continue using *initime* and *tstart* = 2, warning on inconsistency and continue using data time = else, warning on inconsistency and stop. This option is default
initopt	Option for initializing the time dependent fields.	1 = initialize using analytic functions. 2 = initialize with a restart file from a previous run. 3 = initializing from history format initial condition (IC) data set

pt0opt	Option for specifying the initial potential temperature perturbation when *initopt*=1.	0 = no added initial perturbation. 1 = ellipsoidal bubble (defined below). 2 = random perturbations. 3 = random perturbations symmetric about central *x-z* and *y-z* planes. 4 = half vertical wave length bubble (see below). 5 = Soup-can-shaped potential temperature perturbation. 6=ellipsoidal bubble with perturbation specified in *T* rather than θ.
ptpert0	The magnitude (K) of the initial potential temperature perturbation for *initopt*=1 case and all options of *pt0opt*.	Typically a few degrees. *ptpert0* = θ_0 in next block.
pt0ctrx *pt0ctry* *pt0ctrz* *pt0radx* *pt0rady* *pt0radz*	The center location (m) and the radii (m) of the initial bubble perturbation in *x, y* and *z* direction. For *pt0opt* =1, the ellipsoidal bubble is defined by $$\Delta\theta = \theta_0\, cos^2(\pi\beta/2) \quad (for\ \beta \le 1)$$ where θ_0 (= *ptpert0*) is the amplitude at the center of the disturbance and β is a non-dimensional radius given by: $$\beta = \sqrt{\left(\frac{x - x_c}{x_r}\right)^2 + \left(\frac{y - y_c}{y_r}\right)^2 + \left(\frac{z - z_c}{z_r}\right)^2}$$ where x_c = *pt0ctrx*, y_c = *pt0ctry*, z_c = *pt0ctrz* x_r = *pt0radx*, y_r = *pt0rady*, z_r = *pt0radz* For *pt0opt*=4, a 2-D bubble is defined by: $$\Delta\theta = \theta_0\, \frac{sin(\pi z/H)}{1+(x-x_c)^2/x_r^2}$$ where $H = (nz-3)*\Delta\zeta$ (*dz* in the code).	To place the bubble at the center of model domain, set *pt0ctrx = (nx-3)*Δx/2, pt0ctry = (ny-3)*Δy/2.* A negative value of radius implies an infinite extent of the bubble in that direction. Therefore, setting *pt0rady* = -1 gives a bubble perturbation independent of *y*.

sndfile	Name of the sounding file used to initialize the model base state variables when *initopt*=1 and *inibasopt=1*. A sounding file must be defined when *initopt*=1. The sounding file format is defined in Section 8.4.1. CHARACTER (LEN=80) :: *sndfile*.	Character string in quotes. Not to exceed 80 characters in length. *e.g., sndfile='may20.snd'.*
rstinf	Name of the restart file used only when *initopt*=2. The restart file should always be in the machine native binary format. CHARACTER (LEN=80) :: *rstinf*.	Character string in quotes. *e.g., rstinf='arpstest.rst003600'.*
inifmt	Format flag for the initial condition data files *inifile* and *inibgf*. Standard ARPS history data dump formats are used by these initial data files although not all formats are available for the initial condition purpose.	1 = unformatted binary. 2 = ASCII. 3 = NCSA HDF Version 4. 10 = GRIB.
inisplited	Option flag indicating whether the input data files are in single joined form or in split form as created by *splitfiles*. Valid for MPI mode and *initopt* = 3 only.	0 = in single joined form. This option only works when the input files are in binary or HDF format. 1 = in split form
inifile	Name of the initial condition data file containing time-dependent variables, and, possibly, the base-state arrays and grid coordinate arrays depending on the values of flags inside the data file. When both the base state and grid coordinate arrays are present in this file, file *inibgf* will not be read. Used only when *initopt*=3. CHARACTER (LEN=80) :: *inifile*.	Character string in quotes. *e.g., inifile='arpstest.bin003600'.*
inibgf	Name of the initial condition data file containing time-independent base state and grid coordinate arrays. Used only when *initopt*=3. CHARACTER (LEN=80) :: *inibgf*.	Character string in quotes. *e.g., inibgf='arpstest.bingrdbas'.*

inibasopt	Option for initializing the base state variables for *initopt=1* case.	1 = initialize the base state using a single sounding. 2 = isentropic atmosphere. 3 = isothermal atmosphere. 4 = atmosphere with a hard-coded constant static stability ($N=10^{-2}$ s^{-1}). 5 = an analytic thermodynamic sounding profile after Weisman and Klemp (1982) (Section 8.4.5). For *inibasopt* $\neq 1$, additional parameters are hardwired inside subroutine INIBASE.
viniopt	Option for specifying the initial and base-state wind profiles internally when *inibasopt* $\neq 1$.	1 = *ubar=ubar0, vbar=vbar0* (see below). 2 = user-specified profiles for *ubar* and *vbar*. To use option 2, a user has to edit subroutine INIBASE. The default values in the code are 0.0 for *ubar* and *vbar*.
ubar0	Constant u-velocity (m/s) for the initial and base-state wind when *inibasopt* $\neq$ 1 and *viniopt* = 1.	User specified.
vbar0	Constant v-velocity (m/s) for the initial and base-state wind when *inibasopt* $\neq$ 1 and *viniopt* = 1.	User specified.
soilinitopt	Option for a period of initialization in which the soil model is integrated for a time period *soiltintv* when keeping the atmospheric state fixed. The procedure tries to establish an initial state of the soil model that is more in balance with the near surface atmospheric variables.	Used when initial conditions of the soil model are poorly known and when initial balance is desired.

| soiltintv | Length of the balancing integration of soil model when *soilinitopt*=1. | Thirty minutes to one hour. |
| tsfcopt | Option for adjusting soil model skin temperature based on radiation and current air temperature. Used by ADAS only and affects skin temperature output from ADAS. | Procedure applied when parameter set to > 0. |

Parameters for Incremental Analysis Update Nudging (&nudging)

Parameter	**Definition/Purpose**	**Options/Suggested Values**
nudgopt	Option for performing incremental analysis update (IAU) nudging (Bloom et al 1996, MWR). This 'nudging' procedure is different from traditional procedure where a nudging turn to added to the right hand side of prognostic equations.	0 = no nudging 1 = nudging applied
ndstart *ndstop*	Start and end times (second) of the nudging window.	Due to complications with the first leap-frog time step, it is recommended that ndstart >= dtbig
ndintvl	Time interval (second) between incremental updates.	Due to the nature of leap-frog integration it is recommended that *ndintvl* be an odd multiple of *dtbig*.
ndgain	Multiplier applied to nudging at each step.	Typically between 1 and 1.2.
incrfnam *incrfmt*	File name and format of the ADAS analysis increment data. CHARACTER (LEN=80) :: *incrfnam*	incrfmt = 1: binary format incrfmt = 3: HDF format.

nudgu *nudgv* *nudgw* *nudgp* *nudgpt* *nudgqv* *nudgqc* *nudgqr* *nudfqi* *nudgqi* *nudgqs* *nudgqh*	Option flags for nudging (applying analysis increment to) individual prognostic variables (e.g., *nudgu* for *u* velocity component).	0 = No. 1 = Yes.

Parameters For Model Terrain Specification (&terrain)

Parameter	**Definition/Purpose**	**Options/Suggested Values**
ternopt	Model terrain option. When *ternopt* =0, terrain height is zero and codes related to terrain formulations are switched off to improve efficiency.	0 = no terrain, ground is flat. 1 = analytic mountain. 2 = terrain data read from terrain data file *terndat*.
mntopt	Option for analytic mountain type. The bell-shaped mountain is given by $$h = \frac{h_m}{1+\left[(x-x_c)/x_r\right]^2 + \left[(y-y_c)/y_r\right]^2}$$ where h_m is the mountain height, and, x_r and y_r are the half-widths in *x* and *y* directions respectively.	1 = Bell-shaped mountain. 2 = User specified mount profile (code change required).
hmount	Maximum height (m) of the analytic mountain above *zrefsfc* level. Used only when *ternopt* = 1. *hmount* = h_m in the above equation.	$0 \le hmount < (nz\text{-}3)*d\zeta = z_{top}$.

	The half-width (m) of the analytic mountain in *x* and *y* directions, respectively. Used only when *ternopt* =1. *mntwidx* = x_r and *mntwidy*=y_r in the above equation.	A negative value indicates infinite extent in that direction, therefore the mountain collapses to 2-D or even 1-D. When the model is run in 2-D or 1-D mode, one or both of these values become irrelevant.
mntwidx *mntwidy*		
mntctrx *mntctry*	The center location (m) of the analytic mountain in *x* and *y* directions, respectively. Used only when *ternopt =1*. *mntctrx = x_c* and *mntctry=y_c* in the above equation.	When the model is run in 2-D or 1-D mode, one or both of these values become irrelevant.
terndta	Name of the terrain data file. Used only by option *ternopt*=2. CHARACTER (LEN=80) :: *terndta*	A character string in quotes. *e.g., terndta = 'arpstern.dat'.*
ternfmt	Integer format flag for terrain data file *terndta*.	1 = binary 3 = HDF

Model Grid Setup Parameters
(&grid)

Parameter	Definition/Purpose	Options/Suggested Values
dx	Grid spacing (m) in ξ-direction ($\Delta\xi$). In the code, *dx* is used to represent $\Delta\xi$ for notational convenience.	Problem dependent.
dy	Grid spacing (m) in η-direction ($\Delta\eta$). In the code, *dy* is used to represent $\Delta\eta$ for notational convenience.	Problem dependent.
dz	Grid spacing (m) in the ζ-direction ($\Delta\zeta$) in computational space. In the code, *dz* is used to represent $\Delta\zeta$ for notational convenience. *dz* is also the average physical spacing in the vertical before the terrain transformation.	Problem dependent.

strhopt	Option for vertical grid stretching.	0 = no stretching. Uniform vertical grid used 1 = cubic function used to define *dz* (Eq. (7.3.5.)). 2 = tanh function used to define *dz* (Eq. (7.3.6.)).
dzmin	Minimum grid spacing (m) in the vertical direction in physical space when vertical stretching is invoked *dzmin* and *dz* are used to construct a vertically stretched grid according to Eqs. (7.3.4). *dzmin* is reset to *dz* if *strhopt*=0.	A value less than or equal to *dz*.
zrefsfc	Reference height (m) of the model bottom boundary. $zrefsfc = z_0$ in Eq. (7.3.2).	When the entire model domain is elevated, *zrefsfc* can be chosen to be the minimum elevation of the terrain height. 0.0 m recommended.
dlayer1 *dlayer2*	*dlayer1* (m) = the depth of layer 1. *dlayer2* (m) = the depth of layer 2. The vertical domain is divided into 3 layers: layer 1 (the lowest level) has a uniform resolution of *dzmin*.; layer 2 (which is just above layer 1) has a grid spacing that stretches from *dzmin* upwards; layer 3 (top layer) has a spacing of approximately that at the top of layer 2. *dlayer1* = D_1 and *dlayer2* = D_2 in Eqs. (7.3.4.). See Fig. 7.3 for further information. Used only when *strhopt*≠0.	$0 \leq dlayer1 < z_{top}-z_0$. $0 \leq dlayer2 < z_{top}-z_0$, $0 \leq dlayer1+ dlayer2 < z_{top}-z_0$, $dlayer2 = min(dlayer2, z_{top}-dlayer1-z_0)$. *dlayer1*=0, *dlayer2*=10^5 m recommended.
strhtune	Tuning factor (α) for stretching option 2. See Eq. (7.3.6).	$0.2 \leq strhtune \leq 5.0$. A larger value gives a more linearly stretched grid. Default value is 1.0.

Parameter	Definition/Purpose	Options/Suggested Values
zflat	The height (m) at which grid levels become flat in the terrain-following coordinate transformation (see Fig. 7.6). Note: This parameter has nothing to do with the setup of stretching.	$h_m < zflat \leq z_{top}$, where h_m is the mountain top height. If one desires the coordinate surface to become flat at the model top, set *zflat* to a value larger than *ztop*, the model will reset *zflat* = *ztop*.
ctrlat *ctrlon*	The latitude (degrees north) and longitude (degrees east) at the center of model domain.	Anywhere on the globe. $-90 \leq ctrlat \leq 90$ $-180 \leq ctrlon \leq 180$

Map Projection Parameters
(&projection)

Parameter	Definition/Purpose	Options/Suggested Values
mapproj	Map projection option. In general, polar stereographic is a better choice for hemispheric or polar applications, Lambert is better for mid-latitudes and Mercator better for tropical applications.	0 = no map projection. 1 = northern hemisphere polar stereographic projection. -1 = southern hemisphere polar stereographic projection. 2 = northern hemisphere Lambert conformal projection. -2 = southern hemisphere Lambert conformal projection. 3 = Mercator projection 4 = longitude-latitude grid (not available for model runs). A projection that has a map factor close to 1 is recommended.
trulat1 *trulat2*	1st and 2nd true latitude (degrees north) of map projection, respectively. *trulat2* is used only by Lambert projection.	True latitudes near the center of model domain are recommended.

trulon	True longitude (degrees east) of map projection. This longitude line corresponds to the *-y* axis of the model grid.	A longitude near the center of the model domain is recommended.
mpfctopt	Option for factor calculation in the model.	0 = map factor is set to 1 (use this option for special tests only) 1 = map factor determined according to the projection chosen.
mptrmopt	Option flag for including additional map projection related terms in the momentum equations.	0 = additional terms not included 1 = additional terms included (recommended option)
maptest	A flag for performing special map projection related tests.	0 = general use 1 = symmetric definition of map factors for testing.

Time Integration Control Parameters
(&timestep)

Parameter	Definition/Purpose	Options/Suggested Values
dtbig	Large time step size associated with the leap-frog time integration of the non-acoustic wave modes. Acoustic waves are integrated inside the small time steps, using a mode splitting technique. When *ptsmlstp=1*, internal gravity waves are also handled in the small time steps. When *ptsmlstp* =0, the linear stability constraint on *dtbig* is $dtbig \leq \min(2/N,$ $$\frac{1}{|V_{max}|}\left[\left(\frac{1}{\Delta x}\right)^2 + \left(\frac{1}{\Delta y}\right)^2 + \left(\frac{1}{\Delta z}\right)^2\right]^{-1/2})$$ where V_{max} is the maximum wind speed, and N is the Brunt-Väisälä frequency. However, when the flow is highly anisotropic, i.e., when $V_h >> w$, the approximate constraint becomes $dtbig \leq \min(2/N,$ $$\frac{1}{|V_{h\ max}|}\left[\left(\frac{1}{\Delta x}\right)^2 + \left(\frac{1}{\Delta y}\right)^2\right]^{-1/2}, \frac{\Delta z}{w_{max}})$$ When *ptsmlstp =1*, *dtbig* becomes independent of N, therefore N disappears from the above inequalities. It is important to note that the mixing (diffusion) processes impose a constraint on the large time step (*dtbig)* size as well. The settings of the mixing parameters are given later in the computational mixing namelist.	It is recommended that *dtbig* is chosen to be approximately 70% of the value suggested by linear theory.

tstart	The starting time (s) of a model run. *tstart* is used only when *initopt*=1 or 3. For *initopt*=2 (model restart), *tstart* is reset to the time of restart data.	For a run starting from an external data set (*initopt*=3), *tstart* should be set so that model time zero matches the real time set by string *initime*. 0.0 is the default value.
tstop	The model time (s) at which time integration is stopped.	*tstop* ≥ *tstart*.

Small Time Step Control Parameters (&acoustic_wave)

vimplct	Option for vertically implicit *w* and *p* equation solver.	0 = solve *w* and *p* equations explicitly. 1 = solve *w* and *p* equations implicitly in the vertical. 1 should be used most of the time.
tacoef	Weighting coefficient (non-dimensional, ND) for the future time level in the vertically implicit small time step *w* and *p* solvers when *vimplct* = 1. Not used when *vimplct*=0. *tacoef* = β in Eqs. (6.2.31c) and (6.2.31d)	$0.5 \leq tacoef \leq 1.0$. When *tacoef*=0.5, the current and future time levels have equal weight. *tacoef*=0.6 is recommended.
csopt	Option for specifying the sound wave speed (*csound*, see below).	1 = sound speed is a function of base-state ($c_s = \sqrt{\gamma R \bar{T}}$, $\bar{T}$ being the base-state temperature). 2 = as in 1, but reduced by a factor *csfactr*. 3 = a constant ($c_s = $ *csound*, see below). 1 is recommended.

csfactr	Factor by which the sound speed is multiplied. Reducing the sound speed can improve the efficiency of the integration, provided that *csound* is not less than approximately twice the speed of the fastest physically-important signal in the flow (nominally internal gravity waves). Used by *csopt* = 2 case only.	*csfactr* ≥ 0.5 recommended.
csound	A constant sound wave speed (m/s) to be used when *csopt* = 3.	Large values (*e.g.*, 800 m/s) will give an incom-pressible limit while much smaller values (100 m/s) will improve the model's efficiency and effectively make the flow super-compressible. A value of 150 m/s or larger is recommended.
ptsmlstp	Option for integrating internal gravity wave modes (the potential temperature equation) inside small time steps. Doing so removes the dependency of stability condition for *dtbig* on static stability.	0 = integrate θ equation in large time steps. 1 = integrate θ equation inside small time steps. If advection instead of internal gravity waves is the most restrictive process on the *dtbig*, option 0 should be chosen. 0 is recommended.

dtsml	Small time step size associated with the forward-backward time integration of the acoustic terms in *u, v, w* and *p* equations, and the gravity wave terms in θ equation when *ptsmlstp*= 1. When *vimplct* = 0, the linear stability constraint for pure sound waves is $$dtsml \leq \frac{1}{c_s}\left[\left(\frac{1}{\varDelta x}\right)^2 + \left(\frac{1}{\varDelta y}\right)^2 + \left(\frac{1}{\varDelta z_{min}}\right)^2\right]^{-1/2}$$ where c_s is the maximum sound wave speed. When *vimplct*=1, the constraint is $$dtsml \leq \frac{1}{c_s}\left[\left(\frac{1}{\varDelta x}\right)^2 + \left(\frac{1}{\varDelta y}\right)^2\right]^{-1/2}.$$ If *ptsmlstp*=1, *dtsml* takes the lesser value of *dtsml* given above and *2/N*.	*dtsml* should be slightly smaller than that suggested by linear theory, and be less or equal to *dtbig*. When *dtbig* is not an exact multiple of *dtsml*, *dtsml* is reduced so that *dtbig* becomes a multiple of *dtsml*.

Options Related to Equation Formulation (&equation_formulation)

buoyopt	Option to include the buoyancy term on and off	0 = buoyancy is turned off. 1 = buoyancy is turned on. It should be on for general purposes.
buoy2nd	Option for including second-order terms in the linearized buoyancy terms.	= 1, including 2nd-order terms; = 0, only the 1st-order terms. Option 1 is recommended.
rhofctopt	Option for using full density in the pressure gradient force (PGF) terms.	= 0, use base state density in PGF terms; = 1, use full density in PGF terms. Option 1 recommended.

bsnesq	Option for making Bousinessq approximation to the equations.	=0, no Bousinessq approximation; =1, with Bousinessq approximation. Option 1 should be used for special purposes only.
peqopt	Option for using an alternative formulation for pressure equation.	= 1, Original formulation as described in ARPS User's Guide. = 2, An alternative formulation for special applications. Option 1 recommended.

Options for Spatial Advection
(&numerics)

Parameter	Definition/Purpose	Options/Suggested Values
madvopt	Option for momentum advection.	=1, second-order advection; =2, fourth-order in horizontal and second-order in vertical advection; =3, fourth-order advection in all directions. Option 3 is most accurate.

sadvopt	Option for scalar advection. Option 4 is Zalesak's multi-dimensional version of FCT (flux-corrected transport) based on second-order or fourth-order centered and first-order upstream schemes. FCT scheme is applied to potential temperature, water variables and TKE, while either 2nd or 4th order advection is used for pressure depending option *fctorderopt*. Option 5 is a simple positive definite advection (MPDCD) scheme based on flux correction/limiting on leapfrog-centered advective fluxes. With this option, positive definite water variables and TKE are advected using this scheme while potential temperature and pressure are advected by either 2nd or 4th-order centered scheme (i.e., corresponding to *sadvopt*=1 or 3) depending option *fctorderopt*.	1 = second-order advection. 2 = fourth-order in horizontal and second-order in vertical advection; =3, fourth-order centered-in-space advection in all directions. = 4, FCT scheme = 5, simple positive definite advection (MPDCD) scheme. The most accurate (also most expensive) choices are: *madvopt*=3, *sadvopt*=4, *fctorderopt*=2 with *fctadvptprt*=1. The most econimical choices are: *madvopt*=1, *sadvopt*=1. Option 3 is most commonly used although 5 is a better choice.
fctorderopt	Option of the spatial order of accuracy of the high-order scheme used to construct FCT advection (*sadvopt*=4) and MPDCD advection schemes (*sadvopt*=5).	= 1, 2nd order = 2, 4th order Option 2 is more accurate.
fctadvptprt	Option for FCT advection for potential temperature (when *sadvopt=4*).	= 0, FCT scheme is applied to *ptbar+ptprt*. Not recommended. = 1, FCT scheme is applied to *ptprt* only. This option is RECOMMENDED! = 2, FCT scheme is applied to *ptbar+ptprt*-min(*ptbar+ptprt*).

Boundary Condition Parameters
(&boundary_condition_options)

Parameter	Definition/Purpose	Options/Suggested Values
lbcopt	Option for lateral boundary conditions (LBC).	1 = internally determined LBC including rigid wall, periodic, zero-gradient, wave-radiating and two-way interactive nesting. 2 = externally forced LBC. In this case, ebc, wbc, nbc, and sbc will be reset to 5 during initialization.
wbc, ebc nbc, sbc	Lateral boundary options on the west (*wbc*), east (*ebc*), north (*nbc*), and south (*sbc*) sides of the domain. When the model grid collapses to 2-D or 1-D, the periodic condition is enforced in the appropriate direction(s). When *lbcopt* = 2, *wbc, ebc, nbc* and *sbc* are automatically set to 5. When *lbcopt* = 1, they cannot be set to 5.	1 = rigid wall. 2 = periodic. 3 = zero normal gradient. 4 = wave-radiating (see also *rbcopt*). 5 = externally specified time- dependent condition. 6 = interactive nesting (adaptive grid refinement interface needed).

rbcopt	Option for wave-radiating lateral boundary conditions (RBC). Note: These conditions are applied to horizontal velocities u and v ONLY. Option 4 works well most of the time.	1 = Klemp and Wilhelmson type radiation condition that uses user-specified constant phase speed (see *c_phase* below). Radiation condition is computed and applied on the small time step. 2 = Klemp and Wilhelmson type radiation condition that uses user specified constant phase speed (see *c_phase* below). Computed on the big time step and applied on the small time step. 3 = Orlanski (1976) type radiation condition that uses locally diagnosed phase speed for *u* and *v*. Computed on the big time step and applied on the small time step. 4 = Klemp - Lilly/Durran (1983) type radiation condition in which the phase speed is vertically averaged. This is computed on the big time step and applied on the small time step.
c_phase	Constant phase speed (m/s) used in *rbcopt*= 1 and 2 to radiate internal wave signals out of the computational domain.	A value between 30 and 300 m/s for *rbcopt=1* and a value between 30 and 45 for *rbcopt*=2 is recommended.
rlxlbc	Relaxation coefficient (ND) used by radiation boundary condition options 1, 2 and 3. When *rbclbc* ≠ 0, model prognostic variables are weakly relaxed towards their base state values at inflow (in terms of wind speed) boundaries. The amount of relaxation is proportional to the local Courant number.	$0.0 \le rlxlbc \le 0.5$. 0.05 recommended.

bbc	Options for the bottom boundary condition.	1 = rigid lid or rigid ground. 2 = periodic (both top and bottom are set to periodic). 3 = zero normal gradient. Important note: If *vimplct*=1, only option 1 is available.
tbc	Options for the top boundary condition. Option 4 requires a statically stable base state at scalar nz-2. It will run with a neutral environment but the accuracy (and application of the condition) is questionable. In addition, *zflat* must be set to a level at or below the scalar point *nz-3*.	1 = rigid lid or rigid ground. 2 = periodic (both top and bottom are set to periodic). 3 = zero normal gradient. 4 = Linear hydrostatic radiation top boundary (Klemp and Durran MWR, 1983; Chen MWR, 1991). If *vimplct*=1, only options 1 and 4 are valid.
fftopt	Option for Fast Fourier Transform method to use with the upper boundary *tbc*=4. NOTE: The simulation will NOT stop if *nx* or *ny* is improperly chosen to satisfy the FFT constraints. In this case the Fourier transform will not use FAST Fourier transform algorithm.	= 1, periodic transform used, edges are assumed to be equal in value. This option requires special dimensions for (nx,ny) given by, $nx\text{-}1 = 2^p * 3^q * 5^r$, $ny\text{-}1 = 2^p * 3^q * 5^r$ where $p \geq 1$, $q \geq 0$, and $r \geq 0$. *nx* and *ny* must be odd numbers unless it is equal to 4 (for 2D runs). = 2, use even cosine transform, edges are not assumed to be equal in value. For fast cosine transform, add one to the *nx* and *ny* that satisfy the earlier equation, i.e., $nx = 2^p * 3^q * 5^r$, $ny = 2^p * 3^q * 5^r$. The result is that nx and ny must be even!

pdetrnd	Option to turn on pressure detrending procedure. With the option on, the domain averaged perturbation Exner function is reset to zero every time step to remove domain-wide pressure drift/trend sometimes seen when open boundary condition is used.	0 = pressure detrending off. 1 = pressure detrending on. 0 recommended. The detrending SHOULD NOT be used when LBC is provided externally (*lbcopt*=2).

Control Parameters for External Boundary Condition (&exbcpara)

Parameter	**Definition/Purpose**	**Options/Suggested Values**
exbcnam	A prefix of the names of the external boundary condition files. These files should be named as *exbcnam.yyyymndd.hhmmss*, where string *yyyymndd.hhmmss* identifies the year, month, day, hours, minutes and seconds of the data, respectively CHARACTER (LEN=80) :: *exbcnam*.	String in quotes. Not to exceed 80 characters in length. *e.g., exbcnam =* 'arpsexbc'.
exbcfmt	Format of external boundary data files.	= 1, binary (default); = 3, HDF (uncompressed or compressed).
tinitebd	A string identifying the time of the first external boundary condition file. The format is *yyyy-mn-dd.hh:mm:ss*, where *yyyy* , *mn*, *dd*, *hh, mm* and *ss* are integers represent year, month, day, hours, minutes and seconds, respectively. CHARACTER (LEN=19) :: *tinitebd*.	String in quotes of length 19. *e.g., tinitebd='1970-01-01.03:00:00'.*
tintvebd	Time interval (s) at which the existence of external boundary condition files is checked.	Chosen so that all available boundary files can be read.
ngbrz	The number of grid zones inside the boundary relaxation zone for the Davies type externally forced boundary condition (*lbcopt = 2). nbrlx = n_b* in Eq. (8.6.2).	5-7 points recommended. A larger value can be used if the domain is large (>100 grid points).

brlxhw	The half-width of the boundary relaxation function in terms of the number of grid zones. *brlxhw* = b in Eq. (8.6.2)	$0 < brlxhw \leq ngbrz$. A value slightly less than *ngbrz*/2 is recommended.
cbcdmp	The maximum coefficient (s^{-1}) of Rayleigh type damping in the boundary relaxation zone. *cbcdmp* = K_{b0} in Eq. (8.6.2). $\tau = 1/$ *cbcdmp* is the e-folding damping time scale.	Problem dependent. Values corresponding to τ between 10 to 20 *dtbig* generally work well.
cbcmix	The maximum coefficient (s^{-1}) of additional second-order horizontal computational mixing in the boundary relaxation zone. *cbcmix* is K_{c0} and is described in further detail in Section 8.6.	Similar to the second-order computational mixing coefficient. Although most of the time, additional mixing is unnecessary.

Coriolis Force Control Parameters
(&coriolis_force)

Parameter	**Definition/Purpose**	**Options/Suggested Values**
coriopt	Option for including the effect of Earth's rotation. In the equations of motion, the Earth's rotation is defined as follows: $$\left(\frac{du}{dt}\right)_{Coriolis} = f\,\hat{v} - \tilde{f}\,w$$ $$\left(\frac{dv}{dt}\right)_{Coriolis} = -f\,\hat{u}$$ $$\left(\frac{dw}{dt}\right)_{Coriolis} = \tilde{f}\,u$$ where $f = 2\Omega\,sin(\phi)$, $\tilde{f} = 2\Omega\,cos(\phi)$, and Ω = Earth's rotation rate. ϕ is the Earth latitude. When *coriotrm*= 1, $\hat{u}=u$, and $\hat{v}=v$. When *coriotrm*= 2, $\hat{u}=u-\overline{u}$, and $\hat{v}=v-\overline{v}$.	0 = no Coriolis effect. 1 = Only Coriolis terms related to horizontal wind (those involving f). f is evaluated at the model central latitude. 2 = Coriolis terms related to vertical wind (those involving $\tilde{f}$) also included. f and $\tilde{f}$ are evaluated at the model central latitude. 3 = option 1, but f is a function of latitude and effects of spatial gradient of map factor and of earth curvature are also included 4 = option 2, but f and $\tilde{f}$ are functions of latitude and effects of spatial gradient of map factor and of earth curvature are also included
earth_ curvature	Option flag for including terms due to earth curvature effect (see Xue et al. 2000, ARPS Part I) when coriopt=3 or 4.	0 = these terms are neglected 1 = these terms are included when coriopt=3 or 4.

coriotrm	An option for imposing an approximate geostrophic initial balance between the base state winds and the pressure gradient force. In the equations of motion, the Earth's rotation enters as follows: $$\left(\frac{du}{dt}\right)_{Coriolis} = f\,\hat{v} \,-\, \tilde{f}\,w$$ $$\left(\frac{dv}{dt}\right)_{Coriolis} = -\,f\,\hat{u}$$ $$\left(\frac{dw}{dt}\right)_{Coriolis} = \tilde{f}\,u$$ where $f = 2\Omega\sin(\phi)$, $\tilde{f} = 2\Omega\cos(\phi)$, and Ω = Earth's rotation rate. ϕ is the Earth latitude. When *coriotrm*= 1, $\hat{u} = u$, and $\hat{v} = v$. When *coriotrm*= 2, $\hat{u} = u - \overline{u}$, and $\hat{v} = v - \overline{v}$. This option is not used if *coriopt* = 0.	1 = Geostrophic balance is not imposed 2 = Geostrophic balance is imposed initially. This option is viable only when the model is initialized with horizontally homogeneous pressure (from e.g., a single sounding) and non-zero base-state wind that is assumed to be in a geostrophic balance.

Subgrid Scale Turbulent Mixing Parameters (&turbulence)

Parameter	**Definition/Purpose**	**Options/Suggested Values**
tmixopt	Option for physical (subgrid scale - SGS) turbulent mixing.	0 = no SGS turbulence. 1 = first-order SGS closure with constant eddy mixing coefficient, *tmixcst* (see below). 2 = Smagorinsky/Lilly SGS diagnostic parameter-ization. 3 = Option 2 + a constant mixing coefficient given by *tmixcst* (see below). 4 = 1.5-order TKE closure. 5 = Germano scheme (not yet available in this version). Option 4 is recommended.

trbisotp	Option for isotropic subgrid scale turbulence.	0 = the turbulence is assumed to be anisotropic, recommended when dx>>dz. 1 = the turbulence is assumed to be isotropic Use when dx~dz.
tkeopt	Option for 1.5 order TKE formulation used by tmixopt=4 Options 1 and 2 represent subgrid scale (SGS) turbulence mixing. Option 3 represents convective boundary layer mixing parameterization which is necessary when horizontal grid resolution is not sufficient to resolve boundary layer eddies. See Xue et al (1996 NWP conf. paper) for discussion.	1 = after Moeng and Wyngaard (default); 2 = after Deardroff; 3 = after Sun and Chang (1986) J.Climate Appl. Meteor.
trbvimp	Option for implicit treatment of vertical turbulence mixing.	0 = explicit; 1 = implicit. This option must be used when *tmixopt*=4 and *tkeopt*=3 for stability.
alfcoef	Time average weighting coefficient (for past time level) used in the vertically implicit mixing.	When *trbvimp*=0, *alfcoef* is is reset to 1. Otherwise, 0.25 is recommended.
prantl	Turbulent Prandtl number (ND) which equals K_m/K_H where K_m and K_H are, respectively, the eddy turbulent mixing coefficients of momentum and heat.	Values of 1 to 1/3 are typically used.

tmixcst	Constant background mixing coefficient (m^2/s). It is added to the diagnosed mixing coefficient and is used only when *tmixopt* = 1 or 3. When *trbisotp*=1, it is used as in $$tmixcst\left(\frac{\partial^2 A}{\partial x^2}+\frac{\partial^2 A}{\partial y^2}+\frac{\partial^2 A}{\partial z^2}\right);$$ When *trbisotp*=0, it is used as in $$tmixcst\left(\frac{\Delta_H^2}{\Delta^2}\left[\frac{\partial^2 A}{\partial x^2}+\frac{\partial^2 A}{\partial y^2}\right]+\frac{\Delta_V^2}{\Delta^2}\frac{\partial^2 A}{\partial z^2}\right)$$ *A* is the variable being mixed. $\Delta=(\Delta x\Delta y\Delta z)^{1/3}$, $\Delta_H=(\Delta x\Delta y)^{1/2}$ and $\Delta_V=\Delta z$. Note: constant turbulent mixing is applied in the physical space, whereas the computational mixing, described below, is applied in the computational space.	The linear stability of the mixing term requires that $$K\frac{\Delta t}{\Delta^2}<\frac{1}{8}$$ where *K* is a dimensional mixing coefficient. *tmixcst* is usually chosen so that the e-folding time of damping on the 2Δ waves [τ =$\Delta^2/(\pi^2\,tmixcst)^{-1}$] is sufficient for removal of short waves from the solution. This background mixing is not necessary when computational mixing is activated.
tmixvert	Option for using simpler vertical-only formulation of turbulent mixing terms. Vertical-only option can be used for runs with dx >~ 20 km.	= 0, full 3D turbulence formulation. = 1, horizontal components are neglected to save computational time.
kmlimit	An upper limit imposed on the turbulent mixing coefficient for numerical stability. *kmlimit* =1 corresponds to the approximate maximum allowable value of *km* for stable numerical integration.	0 < *kmlimit* ≤ 1.0. A value less than 1 is often necessary when the grid aspect ratio is large. 1.0 recommended.

Spatial Computational Mixing Parameters
(&computational mixing)

Parameter	Definition/Purpose	Options/Suggested Values
cmix2nd	Switch for second-order computational mixing. Different from the turbulence mixing, the computational mixing is applied in the computational space along the computational grid lines. It is entirely computational in nature and acts to damp small-scale numerical noise.	0 = turned off. 1 = activated.
cfcm2h	Coefficient (s^{-1}) of second-order computational mixing in the horizontal. The mixing term enters the RHS of the equations as $$\Delta_h^2\, cfcm2h \left(\frac{\partial^2 A}{\partial \xi^2} + \frac{\partial^2 A}{\partial \eta^2} \right)$$ where $\Delta_h = \sqrt{\Delta\xi\Delta\eta}$ is the horizontal grid scale. *cfcm2h* is related to the K_{2H} and α_{2H} of chapter 6 by, $cfcm2h = K_{2H}/\Delta_h^2$, and $\alpha_{2H} = cfcm2h\,\Delta t$. Used only when $cmix2nd \neq 0$	Linear stability analysis requires that $cfcm2h\ \Delta t < 1/8$ ($\Delta t = dtbig$). Use a value just large enough to eliminate the shortest (2Δ) waves. The corresponding e-folding time should be at least 10 *dtbig*. Larger values can cause computational instability!
cfcm2v	Coefficient (s^{-1}) of second-order computational mixing in the vertical. The mixing term enters the RHS of the equations as $$\Delta_v^2\, cfcm2v\, \frac{\partial^2 A}{\partial \zeta^2}$$ where $\Delta_v = \Delta\zeta$ is the vertical grid scale. *cfcm2hv* is related to the K_{2v} and α_{2v} of chapter 6 by, $cfcm2v = K_{2v}/\Delta_v^2$, and $\alpha_{2v} = cfcm2v\Delta t$. Used only when $cmix2nd \neq 0$.	Linear stability analysis requires that $cfcm2v\ \Delta t < 1/8$ ($\Delta t = dtbig$). Use a value just large enough to eliminate the shortest (2Δ) waves. The corresponding e-folding time should be at least 10 *dtbig*. Larger values can cause computational instability!

cmix4th	Switch for fourth-order computational mixing. Different from the turbulence mixing, the computational mixing is applied in the computational space along the computational grid lines. It is entirely computational in nature and acts to damp small-scale numerical noise.	0 = turned off. 1 = activated. 4th-order mixing is more scale selective and is preferred over the 2nd-order mixing.
cfcm4h	Coefficient (s^{-1}) of fourth-order computational mixing in the horizontal. The mixing term enters the RHS of the equations as $$-\Delta_h{}^4\, cfcm4h \left(\frac{\partial^4 A}{\partial \xi^4} + \frac{\partial^4 A}{\partial \eta^4} \right)$$ where $\Delta_h = \sqrt{\Delta\xi\Delta\eta}$ is the horizontal grid scale. *cfcm4h* is related to the K_{4H} and α_{4H} of chapter 6 by, $cfcm4h = K_{4H}/\Delta_h{}^4$, and $\alpha_{4H} = cfcm4h\,\Delta t$. Used only when $cmix4th \neq 0$	Linear stability analysis requires that $cfcm4h\,\Delta t < 1/8\ (\Delta t = dtbig)$. Use a value just large enough to eliminate the shortest (2Δ) waves. The corresponding e-folding time should be at least 10 *dtbig*. Larger values can cause computational instability!
cfcm4v	Coefficient (s^{-1}) of fourth-order computational mixing in the vertical. The mixing term enters the RHS of the equations as $$-\Delta_v{}^4\, cfcm4v\, \frac{\partial^4 A}{\partial \zeta^4}$$ where $\Delta_v = \Delta\zeta$. is the vertical grid scale. *cfcm4v* is related to the K_{4v} and α_{4v} of chapter 6 by, $cfcm4v = K_{4v}/\Delta_v{}^4$, and $\alpha_{4v} = cfcm4v\,\Delta t$. Used only when $cmix4th \neq 0$	Linear stability analysis requires that $cfcm4v\,\Delta t < 1/8\ (\Delta t = dtbig)$. Use a value just large enough to eliminate shortest (2Δ) waves. The corresponding e-folding time should be at least 10 *dtbig*. Larger values can cause computational instability!
scmixfctr	Reduction factor of the computational mixing coefficients for scalars relative to those of velocities. The c-mixing coefficients are multiplied by a factor of *scmixfctr* for scalars.	*0 ≤ scmixfctr ≤ 1.0* Default 1.0

cmix_opt	Option to apply monotonic computational mixing (4th and 6th order only)	0 = no application of monotonic scheme (default) 1 = monotonic applied to 4th order computational mixing 2 = no monotonic, but 6th order computational mixing 3 = monotonic applied to 6th order computational mixing

Divergence Damping Parameters
(&divergence_damping)

Parameter	**Definition/Purpose**	**Options/Suggested Values**
divdmp	Switch for divergence damping. Damping of 3-D divergence fields suppresses sound waves. It has little effect on meteorologically significant wave modes.	0 = damping turned off. 1 = isotropic divergence damping activated (use when $\Delta x \sim \Delta z$); 2 = anisotropic divergence damping activated (use when $\Delta x \gg \Delta z$).
divdmpndh *divdmpndv*	Coefficients of divergence damping in horizontal and vertical momentum equations, respectively. The damping term has the form $$\frac{\partial(\overline{\rho}\vec{V})}{\partial t} \sim divdmpndh\frac{\Delta x \Delta y}{\Delta \tau}\nabla_h D$$ $$+ divdmpndv\frac{(\Delta z)^2}{\Delta \tau}\frac{\partial D}{\partial z}\vec{k}$$ on the right hand side of the momentum equations, where $\Delta \tau = dtsml$. Used only when $divdmp \neq 0$.	< 0.75. (0.05 recommended).

Upper Level Rayleigh Damping Parameters
(&rayleigh_damping)

Parameter	Definition/Purpose	Options/Suggested Values
raydmp	Option for Rayleigh sponge damping at the upper levels of model domain. Rayleigh damping is applied to all perturbation fields except pressure.	0 = damping not used. 1 = damping activated. 1 recommended.
cfrdmp	Maximum Rayleigh damping coefficient (s^{-1}) at the top of the model domain. The damping coefficient is given by: $$\frac{cfrdmp}{2}\left[1 - \cos\left(\pi\frac{z - z_{lower}}{z_{top} - z_{lower}}\right)\right]$$ where z_{lower} is the height (m) of the bottom of the sponge (damping) layer. z_{top} is the height (m) of the top of model domain. Used only when $raydmp \neq 0$.	$cfrdmp^{-1}$ is the e-folding time for damping the perturbations at the model top. It is usually chosen so that vertical gravity waves propagating from the bottom of the damping layer are effectively damped before they reach the top lid. Typically $cfrdmp^{-1}$ is on the order of 20 large time steps.
zbrdmp	The bottom height (m) of the Rayleigh sponge layer. It is z_{lower} in the above equation. Used only when $raydmp \neq 0$.	A sponge layer depth that is 1/3 or more of the total domain depth is recommended.

Time Filter Coefficient
(&asselin_time_filter)

Parameter	Definition/Purpose	Options/Suggested Values
flteps	Coefficient (ND) of Asselin time filter for damping the computational mode associated with the leapfrog scheme used for the large time step integration.	Values between 0.05 to 0.1 recommended. Note: too small a value (*e.g.* 0.001) may damp the physical model excessively. 0.05 recommended.

Moist Processes
(&microphysics)

Parameter	Definition/Purpose	Options/Suggested Values
moist	Option for moist processes.	0 = all processes relating to moisture are turned off. 1 = moist processes included
subsatopt	Option for allowing condensation to occur below 100% relative humidity (*RH*) is required. This option is most appropriate for synoptic scale grid resolutions (>20 km) For option 2, the *RH* when condensation occurs (*rhsat*) is defined as a linear function of *dx* between *dx_rhsatmin* and *dx_rhsat100*, and is *rhsatmin* for *dx>=dx_rhsatmin* and 1.0 for *dx<=dx_rhsat100*, i.e., *rhsat* = max(*rhsatmin*, min(1.0, *rhsatmin*+(1.0-*rhsatmin*)*(dx -*dx_rhsatmin*)/ (*dx_rhsat100-dx_rhsatmin*))).	0 = Condensation occurs only when *RH* >=100% 1 = Condensation occurs when *RH* >= *rhsat*, which is specified below. 2 =The *RH* when condensation occurs is defined as a function of *dx*. See details on the left.
rhsat	Threshold of RH for condensation used by option *subsatopt*=1.	
rhsatmin	Used when *subsatopt*=2. Minimum threshold of *RH* for condensation to occur for a grid interval of *dx_rhsatm*.	
dx_rhsatmin	Used when *subsatopt*=2. The physical grid distance (m) for condensation to occur when *RH=rhsatmin*.	Suggestedvalues: *rhsatmin*=0.85 *dx_rhsatmin*=50000
dx_rhsat100	Used when *subsatopt*=2. The physical grid distance (m) for condensation to occur when RH=100%.	Suggested value: *dx_rhsat100*=5000.

mphyopt	Option for microphysics parameterizations.	0 = microphysics turned off, only saturationa and condensation processes remain 1 = Kessler warm rain microphysics 2 = 6-category water/ice microphysics of Lin et al (1983 JAM). 3 = Schultz (MWR 1995) simple ice microphysics. When *moist*=1 and *mphyopt*=0, only saturation and condensation processes remain with the warm rain option.
impfallopt	Option for treating the hydrometeor sedimentation terms using implicit advection scheme instead of explicit scheme.	0 = explicit scheme 1 = implicit scheme
fallvalpha	Weighting coefficient for time level n+1 in implicit scheme for hydrometeor sedimentation terms. Used only when *impfallopt*=1.	= 0.0, no weighting for the future time level, the scheme becomes explicit. =1.0, no weighting for the current time level. 0.5 is recommended for 2nd-order accuracy.
fallopt	Option for selecting different formulation for the fall velocities of hydrometeors.	1 = Lin et al (1983) fall velocity formulation 2 = Ferrier (1994) formulation with updated coefficients.

cnvctopt	Option for convective cumulus parameterization. Cumulus parameterization can be used together with microphysics parameterization.	0 = no cumulus parameterization; 1 = Kuo scheme with its own grid-scale condensation; 2 = Kuo scheme and Kessler warm rain microphysics; 3 = Kain and Fritsch cumulus parameterization 4 = Betts-Miller-Janjic cumulus parameterization 5 = new version of Kain-Fritsch scheme Kuo scheme is not recommended for high-resolution modeling. The new K-F option is currently most popular choice.
confrq	Time interval (s) between cumulus parameterization updates. Used when *cnvctopt ≠0* only.	From 10 minutes to 1 hour depending on problem scale.
wcldbs	Vertical motion (m/s) required at cloud base for parameterized convection to occur – a parameter used by Kuo scheme only. Used when *cnvctopt = 1* only.	Scale dependent.
idownd	Option to include downdraft in Kuo scheme. Used when *cnvctopt = 1* only.	0 = no downdraft. 1 = a simple downdraft model. 1 recommended.
qpfgfrq	Frequency of grid-scale precipitation updates in seconds in Kuo scheme. Used when *cnvctopt = 1* only.	A value of 120s often used.

kffbfct	A parameter used by the Kain-Fritsch scheme, for specifying the fraction of convectively generated (that generated by cumulus parameterization) condensate to feed back/transfer to grid-resolved rainwater (or snow) field.	=0.0, no feed back; =1.0, feed back all condensate. In this case, all precipitation will be grid-scale. 0.0 < *kffbfct* <= 1.0 recommended when horizontal grid spacing is less than 25km.
kfsubsattrig	Option for turning on sub-saturation in the Kain-Fritsch scheme.	0 = off 1 = on

Atmospheric Radiation Parameterization (&radiation)

radopt	Option switch for radiation parameterization. Notes: 1) When *sfcphy*=3 or 4, and *radopt* = 0, *radopt* will be reset to 1 so that the surface energy balance for soil model can be determined.	0 = No radiation physics considered. 1 = Simplified radiation parameterization that calculates only the radiative forcing at the ground surface. The radiative forcing to the atmospheric is neglected. 2 = The NASA atmospheric radiation transfer parameterization. 3 = Radiative forcing at the surface is read in from OASIS observations. Usually option 2 should be used. Option 3 is for special forced runs only.

radstgr	Option for computing radiation forcing at staggering grid points (every other points) for *radopt* = 2 option. For example for *nx*=*ny*=9, the directly calculation are performed at the "*x*" points, then calculate radiation variables at "o" by averaging from their surrounding "*x*" points. (The "." points are not updated since they are unused for scalar variables).	0 = No staggering; 1 = Staggered; Radiation calculation on x-y plane is at (even,even) and (odd,odd) points. The values at (even,odd) (odd,even) points are averaged from the surrounding four points. This approach reduces the amount of computations.

```
            j
          9 | . . . . . . . . .
          8 | o x o x o x o x .
          7 | x o x o x o x o .
          6 | o x o x o x o x .
          5 | x o x o x o x o .
          4 | o x o x o x o x .
          3 | x o x o x o x o .
          2 | o x o x o x o x .
          1 | x o x o x o x o .
            +------------------   i
              1 2 3 4 5 6 7 8 9
```

At the boundary, the zero-gradient is assumed.

Note: When staggered and when either (*nx*-3)/*nproc_x* or (*ny*-3)/*nproc_y* is odd, the MPI version will NOT produce identical results for different number of processors but the difference is small and within the approximation by calculating radiation every other grid point.

rlwopt	Flag for choosing the longwave radiation options.	0 = transmission functions are computed using the k-distribution method with linear pressure scaling. Cooling rates are not calculated accurately for pressures less than 20 mb. This option is computationally more efficient as compared to option 1. 1 = transmission functions in the co2, o3 in the co2, o3, and the three water vapor bands with strong absorption are computed using table look-up. Cooling rates are computed accurately from the surface up to 0.01 mb.
radshade	Option to take into account the topographic shade effect on shortwave radiation reaching the ground surface.	0 = no topograpghic shade 1 = topographic shade 2 = topographic shade for a idealized valley uniform in the north south direction.
dtrad	Time interval (seconds) between updates of the radiation forcing (used by radopt = 2 only).	5 to 10 minutes often used.
raddiag	Option to dump radiation variables to a file in GrADS format. The frequency is controlled by dtrad (used by radopt = 2 only).	0 = no dump 1 = dump to a file '*runname.radout*' and its GrADAS control file '*runname.radctl*'

Surface Layer Parameterization
(&surface_physics)

Parameter	Definition/Purpose	Options/Suggested Values
sfcphy	Option flag for surface layer fluxes and land-surface/soil-vegetation models.	0 = no surface physics. 1 = surface fluxes calculated from constant drag coefficients, using specified surface values of potential temperature θ and water vapor mixing ratio q_v. 2 = surface fluxes calculated from stability-dependent surface drag coefficients, using specified surface values of θ and q_v. 3 = surface fluxes calculated from constant drag coefficients, using predicted surface values of θ and q_v. 4 = surface fluxes calculated from stability-dependent surface drag coefficients, using predicted surface values of θ and q_v. When *sfcphy* = 3 or 4, a soil-vegetation model is activated.
landwtr	Option distinguishing between land and water in the surface physics calculations. Land and water are distinguished by the vegetation type.	0 = all grid points are treated as land. 1 = distinction made between land and water.
cdhwtropt	Option to use *cdhwtr* instead of calculated values for *cdh* (and *cdq*) over water even for *sfcphy*=2 or 4.	0 = Use calculated values 1 = User specified value *cdhwtr*. Option 0 should be used for general purposes.

cdmlnd ***cdmwtr***	Bulk aerodynamic drag coefficients (ND) for surface momentum fluxes over land and water, respectively. Used only when *sfcphy*=1 or 3. The surface momentum fluxes are defined as $$\left[\bar{\rho}\,\overline{u'w'}\right]_{surface} = -\,\bar{\rho}\,cdm\ \max\,(V,\,V_{min})u$$ $$\left[\bar{\rho}\,\overline{v'w'}\right]_{surface} = -\,\bar{\rho}\,cdm\ \max\,(V,\,V_{min})v$$ where *u* and *v* are the horizontal velocity components evaluated at the lowest grid level above the ground. *V* is the wind speed at that level, and V_{min} is the lower limit of *V* and is set in include file *sfcphycst.inc*. Parameter *cdm* is either *cdmlnd* or *cdmwtr*.	$cdmlnd = 3 \times 10^{-3}$ and $cdmwtr = 1 \times 10^{-3}$ recommended.
cdhlnd ***cdhwtr***	Bulk aerodynamic drag coefficients (ND) for surface sensible heat flux over la and water respectively. Used only when *sfcphy*=1 or 3. The surface sensible heat flux is defined as $$\left[\bar{\rho}\,\overline{w'\theta'}\right]_{surface} = -\,\bar{\rho}\,cdh\ \max(V,\,V_{min})\left(\theta - \theta_g\right)$$ where θ is the potential temperature at the first grid level, *V* is the wind speed at that level, and V_{min} is the lower limit of *V*. The ground temperature θ_g is either user-specified or predicted depending on the value of *sfcphy*. Parameter *cdh* is either *cdhlnd* or *cdhwtr*.	Usually, *cdh* ~ *cdm*. $cdmlnd = 3 \times 10^{-3}$ and $cdmwtr = 1 \times 10^{-3}$ recommended.

cdqlnd **cdqwtr**	Bulk aerodynamic coefficients for surface moisture (latent heat) flux over land and water, respectively. Used only when *sfcphy* = 1 or 3. The surface moisture flux is defined as $$\left[\bar{\rho}\,\overline{w'q_v'}\right]_{surface} = -\,\bar{\rho}\,cdq\,\max(V, V_{\min})\left(q_v - q_{vg}\right)$$ where q_v is the mixing ratio at the first grid level, V is the wind speed at that level, and V_{min} is the lower limit of V. The mixing ratio at the ground surface q_{vg} is either user-specified or predicted depending on the value of *sfcphy*. Parameter *cdq* is either *cdqlnd* or *cdqwtr*.	Usually, $cdq \sim 0.7\ cdm$.
pbldopt	Option for PBL depth determination.. The PBL depth is used when linearly distributing surface fluxes (see below).	1 = PBL depth is user-specified (as *pbldpth0*). 2 = PBL depth is diagnosed from the vertical virtual potential temperature profile. Option 2 is recommended.
pbldpth0	Specified PBL depth (m) for *pbldopt* = 1.	Case dependent.
lsclpbl0	PBL length scale (m) used for *tkeopt*=3.	0.25 recommended by Sun and Chang (1986).
tqflxdis	Option for distributing heat and moisture fluxes quadratically in a specified depth *dtqflxdis*. Use only when near surface vertical resolution is <50m.	0 = no distribution; 1 = distribution over *dtqflxdis* 2 = with distribution over a depth according to similarity 1 recommended.
dtqflxdis	Depth of flux distribution for *tqflxdis* = 1 option.	100 - 200 m recommended.
smthflx	Option to smooth surface fluxes	0 = no smoothing 1 = with smoothing

numsmth	Number of smooth passes. Used when *smthflx*=1.	
sfcdiag	Flag controlling the diagnostic calculations and printing in the surface physics package. The output is generated in a GrADS format and is separate from the ARPS history file.	0 = no diagnostics. 1 = with diagnostics.

Soil-Vegetation Model and Surface Energy Budget (&surface_ebm)

Parameter	Definition/Purpose	Options/Suggested Values
soilmodel _option	Soil model scheme option	1 = Two-layer Force-restore model (Noilhan/Planton scheme) 2 = Multi-layer 'OUSoil' scheme (Based on OSU/NCEP ETA scheme)
nstyp	The number of soil types allowed per grid point.	A value between 1 and 4. >1 types are useful only when soil type data base defines more than one type in each grid cell.
dtsfc	Time step size for integrating the soil model.	*dtsfc =< dtbig*.
soilmodel _forced	Option for performing an offline forced run of the soil model using surface radiation observations.	0 = soil model is run in conjunction with the ARPS atmospheric model 1 = forced mode (Surface data must be available and likely used in conjunction with *soilinit*=5.
dtsfc	Time step size for integrating the soil model.	*dtsfc =< dtbig*.

nzsoil	Number of soil layers.	Maximum number of levels is 100. For *soilmodel_option* =1, *nzsoil*=2.
dzsoil	Averaged vertical grid spacing in transformed computational space (m). Equivalent to the *dz* for the atmospheric component. Used by *soilmodel_option* =2 only.	
zrefsoil	Reference height of the surface (below ground level) (m). Used by *soilmodel_option* =2 only.	
soilstrhopt	Vertical stretching option for multi-layer soil model grid. Used by *soilmodel_option* =2 only. The stretching grid setup uses a procedure similar to that for the atmospheric grid.	0 = no vertical stretching; 1 = vertical stretching with $f=z^3$ function for *dz*. 2 = vertical stretching with hyperbolic tangent function.
soildzmin	Minimum vertical grid spacing near ground surface in physical space (m). Used when *soilstrhopt* = 2 only.	
soildlayer1	Depth (m) of the layer beneath which stretching is not applied. $0.0 =< soildlayer1 < (nzsoil -3)*dzsoil$.	
soildlayer2	Depth of the mid-layer with stretched vertical spacing (m) $0.0 =< dlayer2 < (nz-3)*dz$ and $0.0 =< dlayer1+dlayer2 < (nz-3)*dz$ For consistency, dlayer2 is reset to: min(dlayer2,ztop-dlayer1).	
soilstrhtune	Tuning parameter used when *soilstrhopt* = 2.	A value between 0.2 and 5.0 is recommended. Stretching becomes more linear as *strhtune* increases. Default value is 1.0.

sfcdat	Option for specifying the surface characteristics data. This option is not used when *initopt*=2, i.e., for restart runs. In this case, data in the restart file will be used.	1 = specified using input parameters (given below), 2 = read in from file *sfcdtfl,* 3 = same as option 2, except when *initopt*=3 and the initialization file *inigbf* contains the data arrays, data in *inigbf* will be used instead.

styp	User specified constant soil type (ND). The soil type is based on the USDA definitions. We added two categories for ice and water. Used when *sfcdat*=1 only. <table><tr><td>01</td><td>Sand</td><td></td></tr><tr><td>02</td><td>Loamy sand</td><td>11 17 23</td></tr><tr><td>03</td><td>Sandy loam</td><td>14 20 26 27</td></tr><tr><td>04</td><td>Silt loam</td><td></td></tr><tr><td>05</td><td>Loam</td><td>12 18</td></tr><tr><td>24</td><td></td><td></td></tr><tr><td>06</td><td>Sandy clay loam</td><td>15 21 28</td></tr><tr><td>07</td><td>Silty clay loam</td><td></td></tr><tr><td>08</td><td>Clay loam</td><td>13</td></tr><tr><td>09</td><td>Sandy clay</td><td>19 25</td></tr><tr><td>10</td><td>Silty clay</td><td>16 22</td></tr><tr><td>11</td><td>Clay</td><td>29 30</td></tr><tr><td>31</td><td></td><td></td></tr><tr><td>12</td><td>Ice</td><td>34</td></tr><tr><td>13</td><td>Water</td><td>00</td></tr></table> The numbers on the right hand side above represent Mylne and Henderson-Sellers soil classes.	$1 \le$ *styp* ≤ 13 1 = Sand 2 = Loamy sand 3 = Sandy loam 4 = Silt loam 5 = Loam 6 = Sandy clay loam 7 = Silty clay loam 8 = Clay loam 9 = Sandy clay 10 = Silty clay 11 = Clay 12 = Ice 13 = Water		

vtyp	User specified constant vegetation type (ND). Used when *sfcdat*=1 only.	$1 \leq$ *vtyp* ≤ 14 The vegetation types are classified into 14 categories: 1 = Desert 2 = Tundra 3 = Grassland 4 = Grassland with shrub cover 5 = Grassland with tree cover 6 = Deciduous forest 7 = Evergreen forest 8 = Rain forest 9 = Ice 10 = Cultivation 11 = Bog or marsh 12 = Dwarf shrub 13 = Semidesert 14 = Water
lai0	User specified constant leaf area index (ND). Used when *sfcdat* = 1 only.	*lai 0* ≥ 0.
roufn0	User specified constant surface roughness length (m). Used when *sfcdat* = 1 only.	On the order of 0.01 for land, and 0.001 for water.
veg0	User specified constant vegetation fraction (ND). Used when *sfcdat*=1 only.	Between 0 and 1.
sfcfmt	Format of data file containing surface characteristics.	1 = Fortran unformatted (default); 3 = HDF4 (uncompressed).
sfcdtfl	Name of the file containing the surface characteristics data (soil and vegetation type, leaf area index, surface roughness and vegetation fraction). Used only when *sfcdat* =3. CHARACTER (LEN=80) :: *sfcdtfl*.	String in quotes. Not to exceed 80 characters in length. *e.g., sfcdtfl='arpssfc.data'*.

soilinit	Options for initializing the soil model state variables. When *initopt*=2, values inside the restart file will be used regardless the choice of this option.	1 = specified using input parameters (given below), 2 = read in from soil model data file *sfcinfl*. For variables missing in *soilinfl*, the values in initial file *inifile* will be used when *initopt*=3 and the values in *soilinfl* take precedence over those in *inifile*. 3 = same as option 2, except that the values found in *inifile* take precedence over those found in *soilinfl*. 4 = soil temperature variables are initialized by adding offsets (specified below) to the surface air temperature, while soil moisture variables are initialized from given saturation rates. The canopy water amount is initialized from its default value, *wetcanp0* though. 5 = Soil model variables initialized using Mesonet data. For this option: *radopt* = 3, *sfcfmt* = 2, *soilfmt* = 2.
ptslnd0	User specified constant initial soil surface potential temperature (K). Used only when *soilinit* =1.	Problem dependent.
ptswtr0	User specified constant initial water surface potential temperature (K). Used only when *soilinit* =1.	Problem dependent.
tsoilint(1: *nzsoil*)	User specified initial soil temperatures (K) profile used when *soilinit=1*.	Problem dependent.

qsoilint(1:*nzsoil*)	User specified initial soil moisture (m**3/m**3) used when *soilinit=1*.	Problem dependent.
snowdpth0	Initial snow depth (m). Used only when *soilinit*=1.	Problem dependent.
wetcanp0	User specified constant initial canopy moisture (ND). Used only when *soilinit* =1.	Problem dependent.
ttprt	Offset of top soil temperature from surface air temperature. When *soilmodel_option=1*, *ttprt* will be used as the sfc. temp. offset and *tbprt* will be used as the deepsoil layer temp. offset. When *soilmodel_option*=2, the temp. offset between the top soil layer and the bottom soil layer will be interpolated linearly from *ttprt* and *tbprt*.	Problem dependent.
tbprt	Offset of bottom soil layer temperature from surface air temperature. (See note under *ttprt*)	Problem dependent.
wgrat	Saturation rate of surface soil moisture. Used only when *soilinit*=4.	Problem dependent.
w2rat	Saturation rate of deep layer soil moisture. Used only when *soilinit*=4.	Problem dependent.
soilinfl	Name of the file containing the initial soil model variables (ground surface and deep soil temperature, ground surface soil, deep soil and canopy moisture). Used only when *soilinit* =3. Character *soilinfl*80.	String in quotes. Not to exceed 80 characters in length. *e.g.*, *soilinfl*= '*arpstest.soilinit*'.
soilfmt	Format of data file containing initial values of soil model variables.	1 = Fortran unformatted; 2 = OASIS data; 3 = HDF4 (uncompressed).

Parameter	Definition/Purpose	Options/Suggested Values
tsoil_offset	Option for including seasonal mean deep and skin layer temperature offset in the two-layer force-restore model (*soilmodel_option*=1). See Ren and Xue (2003, manuscript submitted to JAM).	0 = Effect not included 1 = Offset specified as constant (in the following) throughout the domain; 2 =Spatially dependent offset (not implemented yet) If you don't know how to set *tsoil_offset_amplitude*, set this option to 0.
tsoil_offset _amplitude	The amplitude of the annual cycle of the difference (offset) in deep and skin layer soil seasonal-mean temperatures. Generally positive in winter, and negative in summer and near zero in spring and fall.	1 = Fortran unformatted; 2 = OASIS data; 3 = HDF4 (uncompressed).

Automatic Domain Translation Parameters (&grdtrans)

Parameter	Definition/Purpose	Options/Suggested Values
cltkopt	Option for turning on the (storm) cell tracking.	0 = no cell tracking. 1 = cell tracking.
tceltrk	Time interval (s) between calls to cell-tracking algorithm.	120 s is recommended to track storm cells.
tcrestr	Time period (s) during which the mass-weighted cell center is expected to be restored to the domain center. This parameter will only be used when cell tracking is used in conjunction with grid translation (*cetkopt*=1 and *grdtrns*=1).	1800 s recommended for thunderstorm simulations.

grdtrns	Option for automatic domain translation. This procedure adjusts the domain translation speed (*umove, vmove*) periodically to keep the primary features of interest within the computational domain. Note: Parameters *umove* and *vmove* specified earlier will be adjusted during the model run if this option is turned on. The domain translation should not be used if surface features are not homogeneous. Please interpret the model output with caution when domain translation is used.	0 = no automatic translation 1 = use cell-tracking algorithm to estimate the new grid translation speed, and move the mass weighted cell center towards the center of model domain. 2 = use an algorithm that computes the running mean of the optimal domain movement speed and adjusts *umove* and *vmove* (see below) so that primary features remain inside the model grid.
umove *vmove*	Earth-relative translation speed (m/s) of model domain in *x* and *y* directions, respectively. When *inibasopt*=1, (*umove, vmove*) is subtracted from the wind in the sounding. For a run starting from a restart or external data set (*initopt* =2 or 3), the wind field in the data is adjusted so that (*umove, vmove*) is the new domain translation speed. However, when *umove* or *vmove* is 999.0, (*umove, vmove*) in the restart data will be used instead.	Chosen so that the primary feature of interest remains inside the model domain during the course of model integration. When the model solution is related to geographic features, *e.g.* terrain, the domain translation feature cannot be used, *i.e., umove* and *vmove* must be zero.
chkdpth	The domain depth (m) AGL within which interesting features are traced. Used when *grdtrns* = 2 only.	Problem dependent. 2500 m is typically used to track convective cells.
twindow	The time window (s) within which the average domain translation speed is calculated. Used when *grdtrns* = 2 only.	Problem dependent. 300 s is typically used to track convective cells.

Model I/O Control Parameters for History Dumps (&history_dump)

Parameter	**Definition/Purpose**	**Options/Suggested Values**

hdmpopt	Option parameter for specifying the times history dumps are created.	1 = dumps at equal time intervals, start from *tstrdmp*; 2 = dumps at times specified by user.
dmp_out_ joined	Flag indicating if, when the model is run in distributed-memory-parallel mode using MPI, the output fields from different processors will be gathered and joined first before being written out (into single files). NOTE: See the restriction for *max_fopen* flag when *dmp_out_joined* = 1. Joined dumps only work for GrADS, binary and HDF format.	0 = each processor writes out its own portion of data, the output will be joined together using *joinfiles* program. 1 = the output fields from different processors will be gathered and joined first before being written out. The *joinfiles* step is no longer needed.
thisdmp	Time interval (s) between history data dumps when hdmpopt=1. It is reset to the nearest integer multiple of d*tbig* when necessary.	$0 \leq thisdmp$. Setting *thisdmp* to zero switches off history dump file writing.
hdmpfmt	Format flag for ARPS history data dumps. Format 1 is the fastest. Format 3 with compression is most compact and portable. ASCII is portable but the files are very large. Vis5D and GrADS are formats for 3D visualization and GrADAS graphic plotting, respectively. GRIB format will be dropped and the support for NetCDF and HDF 5 support may be added in the future. See Section 10.1 for detailed descriptions.	0 = no history dump. 1 = unformatted binary (whenever available, compiler options are included in *makearps* so that big endian format is used even on little endian platforms such as those using Intel processors). 2 = ASCII. 3 = NCSA HDF version 4. 6 = binary with grid point skipping in partial domain. 7 = NetCDF (not available) 8 = Packed 16-bit NetCDF (not available). 9 = GrADS. 10 = GRIB 11 = Vis5D format.

thisdmp	Time interval (s) between history data dumps when hdmpopt=1. It is reset to the nearest integer multiple of d*tbig* when necessary.	$0 \leq thisdmp$. Setting *thisdmp* to zero switches off history dump file writing.
grbpkbit	Number of bits to use in packing GRIB data	= 32, full precision retained on 32 bit machines; = 16, reduced precision to same disk space. 16 is sufficient for general purposes.
hdfcompr	HDF4 compression option (for *hdmpfmt*=3). It is reset to the nearest integer multiple of d*tbig* when necessary. Note that only options 0-2 work on Cray platforms.	0 = (default), no compression; 1 = fast gzip compression; 2 = high gzip compression; 3 = adaptive or skipping Huffman compression; 4-7 = as above plus mapping reals to 16 bit integers.
tstrtdmp	Time (s) at which history dumps start.	User's choice.
numhdmp	The number of history dump times specified by user for *hdmpopt*=2.	Choose 0 if no history data dump is desired.
hdmptim	Real array, specifying the times when history dumps are created. Used when *hdmpopt*=2. The maximum array length permitted is 100.	

Other Model I/O Control Parameters (&output)

dirname	Name of the directory into which output data are written. CHARACTER (LEN=80) :: *dirname*.	Character string in quotes. Not to exceed 80 characters in length. Default ='./' for current directory.

exbcdmp	Option to write external boundary condition files at the time of history dumps.	0 = no EXBC dump. 1 = EXBC dump in binary format 3 = dumps in HDF format.
exbchdfcompr	EXBC HDF format compression option (for *exbcdmp*=3). Note that only options 0-2 work on Cray platforms.	0 = no compression; 1 = fast gzip compression; 2 = high gzip compression; 3 = adaptive or skipping Huffman compression; 4-7 = corresponding to the above plus mapping real numbers to 16 bit integers.
exdtadmp	Option to write out the history format files that contain the external data arrays interpolated to the current model time. These files will be created at the same time as the history data files.	0 = the data file is not generated. 1 = the data file is created. When $lbcopt \neq 2$, it is automatically reset to 0.
filcmprs	Switch to automatically compress history data files, by using UNIX file compression utility **compress** or **gzip**.	0 = no compression. 1 = compression. Compression can slow down model execution but can reduce the disk usage.
readyfl	Option to create a marker file (same name is the history dump but with "_ready" appended to the end) to indicate the completion of the writing of the history file for the purpose of automatic detection.	0 = do not create a ready file. 1 = create a ready file;
grdout	Option to write out grid coordinate arrays (*x, y, z* and *zp*) together with the time dependent arrays in the history file. Grid arrays are always written into a separate base/grid data file regardless the value of *grdout*.	0 = no. 1 = yes. 0 recommended.

basout	Option to write base state arrays ($\bar{u}$, $\bar{v}$, $\bar{w}$, $\bar{p}$, $\bar{\theta}$ and $\bar{q}_v$) together with the time dependent arrays in the history file. Base state arrays are always written into a separate base/grid data file regardless the value of *basout*.	0 = no. 1 = yes. 0 recommended.
varout	Option to write time dependent model dynamic variables (*u, v, w,* θ and *p*) into the history file. Note that the winds are dumped out in the grid-relative instead of ground-relative framework.	0 = no. 1 = yes. 1 recommended.
mstout	Option to write moisture variable arrays (q_v, q_c and q_r) into the history file.	0 = no. 1 = yes. Set to 0 if the model run is dry. When *mstout*=0, ice variables will not be written either.
iceout	Option to write ice variable arrays (q_i, q_s and q_h) into the history file.	0 = no. 1 = yes. Set to 0 if the model run is dry.
tkeout	Option to write turbulent kinetic energy into the history file.	0 = no. 1 = yes. TKE is non-zero only when *tmixopt* = 4.
trbout	Option to write turbulent mixing coefficient array (k_m) into the history file.	0 = no. 1 = yes.
rainout	Option to write surface accumulated rainfall arrays into the history file.	0 = no. 1 = yes. Set to 0 if the model run is dry.
sfcout	Option to write soil model time-dependent arrays into the history file.	0 = no. 1 = yes. Set to 0 if surface physics is off.

landout	Option to write surface characteristics arrays into the history file.	0 = no. 1 = yes.
prcout	Option to write precipitation rates into the history file.	0 = no. 1 = yes.
radout	Option to write radiation arrays in history dump	0 = no. 1 = yes.
flxout	Option to write surface fluxes in history dump	0 = no. 1 = yes.
qcexout	Option for writing array *qc* in EXBC data dump when *exbcdmp*≠0.	0 = no. 1 = yes.
qrexout	Option for writing array *qr* in EXBC data dump when *exbcdmp*≠0.	0 = no. 1 = yes.
qiexout	Option for writing array *qi* in EXBC data dump when *exbcdmp*≠0.	0 = no. 1 = yes.
qsexout	Option for writing array *qs* in EXBC data dump when *exbcdmp*≠0.	0 = no. 1 = yes.
qhexout	Option for writing array *qh* in EXBC data dump when *exbcdmp*≠0.	0 = no. 1 = yes.
sfcdmp	Option to create surface characteristics data file.	0 = no file generated 1 = binary format 3 = (uncompressed) HDF
soildmp	Option to create soil model initial condition format data files.	0 = no file generated 1 = binary format 3 = (uncompressed) HDF
terndmp	Option to write out an ARPS terrain data file.	0 = no file generated 1 = binary format 3 = (uncompressed) HDF
trstout	Time interval (s) between restart data dumps. Restart data are in the machine native binary format.	It is recommended that restart data be written at the end of a model run and at some intermediate times.

tmaxmin	Time interval between the printouts of domain-wide maximum and minimum values of variables. Output can be plotted by program ARPSPLTMAX.	User discretion.
tfmtprt	Time interval (s) between formatted variable printouts in standard output file.	$0 \le$ *tfmtprt* . Setting *tfmtprt* to zero switches off formatted printing.
tenergy	Time interval between printouts of domain energy statistics (only partially maintained).	User discretion.
imgopt	Option for the generation of 2-D 8-bit raster image files in NCSA HDF format.	0 = no image file. 1 = image files written. The indices of the slices are hardwired in subroutine OUTPUT.
timgdmp	Time interval (s) between creation of HDF-image files.	User discretion.
pltopt	Option for generating graphic plots during model run.	Not implemented.
tplots	Time interval (s) between graphic plot generation.	Not implemented.

Debug Parameters
(&Debug)

Parameter	Definition/Purpose	Options/Suggested Values
lvldbg	Option for printing debug information. Currently ARPS prints certain arrays in tabular form.	0 = no printing. 1= model variables in large time step. 2 = add forcing terms in large time step. 3 = add variables in small time step. 4 = add forcing terms in small time step. 5 = add individual forcing terms and other miscellaneous information.

Parameters Used By Program EXT2ARPS
Gridded Data Processor
(&extdfile)

Parameter	Definition/Purpose	Options/Suggested Values
extdopt	Option for external data sources. Two special definitions: *extdopt* > 50 is reserved for users' definitions. *extdopt* > 100 is reserved to support sub-region GRIB file retrieved through NOMADS (NOAA Operational Model Archive Distribution System, see http://nomad3.ncep.noaa.gov/), *i.e.* the data files are not defined on any existing NCEP grid, but is customized by the users. Currently, only support for GFS 1°×1° global data in grid #3 was implement, *i.e.* *extdopt* = 113, was implemented. Support for other data sources may be added later. NOTEs for *extdopt* > 100: 1. It is users' responsibility to ensure external sub region is at least as large as the ARPS domain; 2. All vertical levels are required; 3. Refer to "*include/gribcst.inc*" for required variables for each external grid.	0 = history dumps from ARPS runs; 1 = NMC RUC (Hybrid-B) data in GRIB (grid #87); 2 = NMC ETA data in GRIB (grid #212); 3 = OLAPS data[*]; 4 = GEMPAK RUC data[*]; 5 = GEMPAK ETA data[*]; 6 = COAMPS data[*]; 7 = NMC RUC AWIPS data in GRIB (grid #211); 8 = NCEP/NCAR T62 Reanalysis 1 data on Gaussian grid in GRIB; 9 = GEMPAK RUC-2 data[*]; 10 = GEMPAK ETA data (grid #104)[*]; 11 = NCEP native coordinate RUC2 in GRIB (grid #236); 12 =NCEP isobaric RUC2 data in GRIB (grid #236); 13 = NCEP AVN/GFS global data in GRIB (grid #3); 14 = NCEP AVN/GFS global data in GRIB (grid #2);

---

[*] Support is still not upgraded to ARPS5.2. Some works may be needed. Used with care.

		15 = NCEP-DOE global Reanalysis 2 (grid #2)[†]; 16 = NMC 12km ETA data in GRIB (grid #218)[‡]; 17 = NARR 32km North American Regional Reanalysis data (grid #221)[§];
extdfmt	Flag for external input data format (used only when ***extdopt*** = 0, *i.e.* for ARPS data files).	see definitions for ***hdmpfmt***.
dir_extd	The directory that contains the external data files to be converted to the ARPS history data format. Character *dir_extd**80.	Character string in quotes. Not to exceed 80 characters in length. For the current directory, set *dir_extd='./'*.
extdname	Prefix string of external file name[**]. Character extdname*80.	Character string in quotes. Not to exceed 80 characters in length. *e.g. extdname='eta'*.
nextdfil	Number of external data files to be converted.	Maximum number is 50. Must be less than or equal to the number of files named by *extdtime*, below.

[†] See http://www.cdc.noaa.gov/cdc/data.ncep.reanalysis2.html for reference.

[‡] See http://www.emc.ncep.noaa.gov/mmb/research/tiles.218.html for reference.

[§] See http://wwwt.emc.ncep.noaa.gov/mmb/rreanl/index.html for reference. Program ***ext2arps*** reads NARR A files and you should also download NARR fixed field file in 32-km output grid.

[**] The actual name of the external file varies with the data sources. However, it is usually constructed based the following parameters, ***dir_extd***, ***extdname*** and ***extdtime***. The template is "***dir_extd/extdname***.*yyyymmddhhfHH*", where *yyyy, mm, dd, hh* and *HH* are extracted respectively from the external time string "***extdtime***".

extdtime	A string array used to specify the UTC time corresponding to the desired external data. The actual name of the external file varies with the data source (*e.g.* NMC RUC or ETA). The string is a concatenation of the initialization time of forecast and the forecast time. It has format '*yyyy-mn-dd.hh:mm:ss+HHH:MM:SS*', where *yyyy, mn, dd, hh, mm* and *ss* are integers for year, month, day, hours, minutes and seconds of the initial time and *HHH,MM,SS* are the forecast time in hours, minutes and seconds. When the data is an analysis rather than a forecast, use *000:00:00* in the forecast time part of the string. Character *extdtime(50)*29*.	Character strings of length 29. *e.g., extdtime(1) = '1970-01-01. 00:00:00+000:00:00'.*
iorder	Option for the polynomial interpolation	1 = Linear interpolation; 2 = Quadratic interpolation; 3 = Cubic interpolation (default).
intropt	Option indicating to interpolate perturbation or total variables	1 = Interpolate perturbation variables and add to base sounding (default); 2 = Interpolate total variables (recommend for *extdopt* = 0).
nsmooth	Number of smoothing passes after interpolation.	1 = Default; 2 = Recommended.
ext_lbc	Option to apply lateral boundary conditions to the winds	0 = no boundary condition applied; 1 = apply zero-gradient boundary conditions (Default).

ext_vbc	Option to apply vertical boundary conditions to *w*	0 = no boundary conditions applied; 1 = apply boundary conditions specifed by **tbc** & **bbc** to *w* (Default).
exttrnopt	Option for terrain on output grid	0 = Use terrain specified by **ternopt** above (Default); 1 = Interpolate terrain from original grid; 2 = Use terrain read in from file but merged to original grid at the boundaries (see also **extntmrg***)*.
extntmrg	Number of zones over which to merge original and new terrain at the boundaries (used only when **exttrnopt** = 2).	Number in grid points. 7 (Default)
extsfcopt	Option for near surface interpolation. (user only when near surface fields are presented, such as *t_2m_ext*, *qv_2m_ext*, *u_10m_ext*, *v_10m_ext*, etc.) Currently only valid for **extdopt** = 2,13,16,17,113, and more work needed for option 7, 12, 51 *etc*.	0 = Do not use near surface fields, instead, use values over pressure levels to interpolate. (Default); 1 = Use near surface fields if they are available.
grdbasopt	Option for writing grid and base history file, which is referred as "*grdbas*" file. This option was added recently for I/O efficiency purpose.	-1 = Never write out the "*grdbas*" files; 0 = Only the first I/O generates "*grdbas*" file (Default). 1 = Always write the "*grdbas*" files. This was the behavior in previous release.

Parameters Used by Program ARPSSFC
Surface Characteristics Data Preprocessor
(&soil_veg_data)

Parameter	Definition/Purpose	Options/Suggested Values
schmopt	Options for the method used to initialize the following surface characteristics: soil type, vegetation type, leaf area index, surface roughness, and vegetation fraction.	0 = constant in the entire domain. 1 = one constant value in a user specified rectangle region (foreground) and another in the rest of the area (background) 2 = constant in the foreground region and data from the surface characteristics database in the background. 3 = data from the surface characteristics database in the entire domain. 4 = Real data for soil type and vegetation types. LAI, vegetation fraction & roughness length are derived using a look-up table (specified by parameter *lkupfl*) based on vegetation types.
sdatopt	Option to select soil texture data sets (Used only when *schmopt* > 1).	1 = STATSGO soil texture data set (1km); 2 = GED soil type data set (1°×1°); 3 = USGS 30" global top layer soil types;

fstypfl /bstypfl	File name of soil classes. Character *vtypfl* *80. *sdatopt* = 1: *fstypfl* specifies the file name of STATSGO data set; *bstypfl* specifies the file name of GED soil type data set. *sdatopt* = 2: Only *bstypfl* is used for the file name of the GED data set; *sdatopt* = 3: Only *fstypfl* is used to specify a directory for 30" global top layer soil files.	Character string for filename or directory name. *e.g, bvtypfl='arpssfc.data /owe14d.data'*
vdatopt	Option to select vegetation data sets (Used only when *schmopt* > 1).	1 = North American 1km vegetation type data set; 2 = GED vegetation type data set (1°×1°); 3 = USGS 30" global landuse data set.
fvtypfl /bvtypfl	File name of vegetation classes *vdatopt* = 1: *fvtypfl* specifies the file name of STATSGO data set; *bvtypfl* specifies the file name of GED vegetation type data set. *vdatopt* = 2: Only *bvtypfl* is used for the file name of the GED data set; *vdatopt* = 3: Only *fvtypfl* is used to specify a directory for 30" global top layer soil files.	Character string for filename or directory name.
ndatopt	Option to select NDVI data sets (Used only when *schmopt* = 2/3)	1 = North American 1km data set; 2 = GED NDVI data set.
fndvifl /bndvifl	File name of vegetation classes Character *ndvipfl* *80. *ndatopt* = 1: *fndvifl* specifies the file name of 1km data set; *bndvifl* specifies the file name of GED NDVI data set. *ndatopt* = 2: Only *bndvifl* is used for the file name of the GED data set;	Character string for filenames. *e.g., ndvifl='arpssfc.data /ndvi.data'*

vfrcopt	Option to select the vegetation fraction data set. (Used only when ***schmopt*** = 2/3)	1 = Default built-in table conversion; 2 = Select NESDIS green vegetation fraction data sets (0.144°×0.144°).
lkupfl	File name of look-up table (for ***schmopt*** = 4 only).	'sfc_winter.tbl'
nsmthsl	Number of smoothing passes applied to *veg* (vegetation fraction), *ln* (roughness length) and *lai* (LAI).	3
drawval	Option for NCARG to draw integer value at each grid point (valid only when ***arpssfc*** was linked with NCARG libraries).	1 = yes, 0 = no, then contour will be draw instead of putting integer values.
fgbgni	Beginning index (*i*) in x-direction of the foreground region. (Used only when ***schmopt*** = 1/2).	User specified.
fgendi	Ending index (*i*) in x-direction of the foreground region.	User specified.
fgbgnj	Beginning index (*j*) in y-direction of the foreground region.	User specified.
fgendj	Ending index (*j*) in y-direction of the foreground region.	User specified.
fgstyp	Soil type for the foreground. (Used only when ***schmopt*** = 1/2).	User specified.
fgvtyp	Vegetation type for the foreground.	User specified.
fglai	Leaf area index for the foreground.	User specified.
fgrfns	Surface roughness for the foreground.	User specified.
fgveg	Vegetation fraction for the foreground.	User specified.

bgstyp	Soil type for the background. (Used only when ***schmopt*** = 1/2).	User specified.
bgvtyp	Vegetation type for the background.	User specified.
bglai	Leaf area index for the background.	User specified.
bgrfns	Surface roughness for the background.	User specified.
bgveg	Vegetation fraction for the background.	User specified.
stypout	Option for including soil type in the surface characteristics data output.	0 = no. 1 = yes. 1 expected.
vtypout	Option for including vegetation type in the surface characteristics data output.	0 = no. 1 = yes. 1 expected.
laiout	Option for including leaf area index in the surface characteristics data output.	0 = no. 1 = yes. 1 expected.
rfnsout	Option for including surface roughness in the surface characteristics data output.	0 = no. 1 = yes. 1 expected.
vegout	Option for including vegetation fraction in the surface characteristics data output.	0 = no. 1 = yes. 1 expected.
ndviout	Option for including NDVI in the surface characteristics data output.	0 = no. 1 = yes. 1 expected.