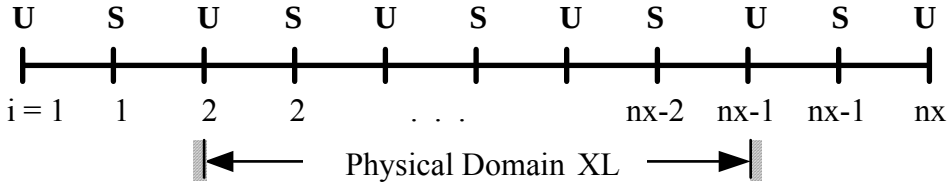
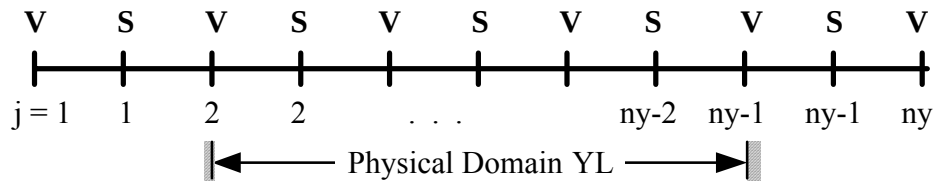
ARPS 5.0 Runtime / Configuration Reference Guide

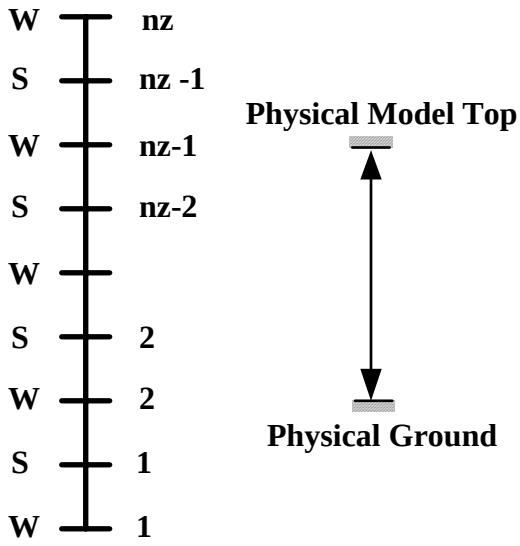
ARPS provides its users with flexible control over many parameters that can be used to configure the model for various applications. Most of these parameters can be set at run time without modifying or recompiling the model code. These parameters are specified in a model input file, *arps.input*. The 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 another word, *arps.input* is shared by a number of programs in the ARPS package. The sharing simplify input file maintenance and avoid 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 groups that are consistent with the 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 $XL = (nx-3)\Delta x$. The grid structure in the x or ξ-direction is:  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 $YL = (ny-3)\Delta y$. The grid structure in the y or η-direction is:  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.

nz	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:  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
nocmnt	Number of comment lines (see definition of *cmnt*).	0≤ nocmnt ≤ 50.
cmnt	Character string array for optional comments to be printed in standard output file. *cmnt*(1) and *cmnt*(2) will be written into the history dump files. Character *cmnt*(50)*80.	Information that will help identify this model run. *e.g., cmnt(1)='Del City Storm'.*

Experiment Identification Parameters (&comment_lines)

Parameter	Definition/Purpose	Options/Suggested Values
nproc_x ***nproc_y***	Number of processors in the *x* and *y* directions, respectively, in the domain-decomposition setup of 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 to use.	Integer values ≥ 1.
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.	Set *max_fopen* = *nproc_x* × *nproc_y* to start with.

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 Lines (&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 50.

Model Run Name (&jobname)

Parameter	Definition/Purpose	Options/Suggested Values
runname	A character string containing the pertinent 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 dumps. This string is also carried in 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, 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) :: *inidate*.	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 = self-initialization using analytic functions. 2 = restarting from 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.
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 $\Delta\theta = \theta_0 \cos^2(\pi\beta/2)$ (*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 input sounding file used to initialize the model base state variables when *initopt*=1 and *inibasopt=1*. A sounding file has to be present 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. Note that NetCDF supported was stopped starting from 4.1.5.	1 = unformatted binary. 2 = ASCII. 3 = NCSA HDF 4. 4 = packed binary. 7 = NetCDF 8 = Packed NetCDF. 10 = GRIB. Formats 5 and 6 are not available for initializtion.

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. INTEGER :: *rstinf*.	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. 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 *iniopt=1* case. The initialization for base state variables is done for all options of *initopt* though. For *initopt* = 2 or 3, the base state arrays initialized this way will be overwritten by those in the data.	1 = initialize the base state using a single sounding. 2 = isentropic atmosphere. 3 = isothermal atmosphere. 4 = atmosphere with a hard-coded constant static stability. 5 = an analytic thermodynamic sounding profile after Weisman and Klemp (1982) (Section 8.4.5). For *inibasopt* ≠1, additional parameters are hardwired inside subroutine INIBASE.

viniopt	Option for specifying the initial and base-state wind profiles internally when *inibasopt* ≠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* ≠ 1 and *viniopt* = 1.	User specified.
vbar0	Constant v-velocity (m/s) for the initial and base-state wind when *inibasopt* ≠ 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 is poorly known and when initial balance is desired.
soiltintv	Length of the balancing integration of soil model when *soilinitopt*=1.	About half 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 turned on parameter set to > 0.

Parameters for Incremental Analysis Update Nudging (&nudding)

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 turned on.
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 to apply 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 needed).
hmount	Maximum height (m) of the analytic mountain above *zrefsfc* level. Used only when *ternopt* = 1. *hmount* = h_m in the above equation.	$0 \le hmount < (nz-3)*d\zeta = z_{top}$.
mntwidx ***mntwidy***	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.
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.
ternfmt	Integer format flag for terrain data file *terndta*.	1 = binary 3 = HDF

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*'.

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. *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 definitional sketch. Used only when *strhopt*≠0.	$0 \le dlayer1 < z_{top}-z_0$. $0 \le dlayer2 < z_{top}-z_0$, $0 \le dlayer1 + dlayer2 < z_{top}-z_0$, $dlayer2 = min(dlayer2, z_{top}-dlayer1-z_0)$. *dlayer1*=0, *dlayer2*=10^5 m recommended.
strhtune	Tuning factor for stretching option 2. It is factor α in Eq. (7.3.6).	$0.2 \le strhtune \le 5.0$. A larger value gives a more linearly stretched grid. Default value is 1.0.
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.	$h_m < zflat \le z_{top}$, where h_m is the mountain top height. If one wants the coordinate surface to bacome flat at the model top lid, one can set *zflat* to a value larger than *ztop*, the model will reset it to *ztop*.
ctrlat ***ctrlon***	The latitude (degrees north) and longitude (degrees east) at the center of model domain.	Anywhere on the globe. $-90 \le ctrlat \le 90$ $-180 \le ctrlon \le 180$

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 choosen.

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.	Should be 0 in general.

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 $dtbig \le \min(2/N,$ $\frac{1}{\lvert V_{\max}\rvert}\left[\left(\frac{1}{\Delta x}\right)^2+\left(\frac{1}{\Delta y}\right)^2+\left(\frac{1}{\Delta z}\right)^2\right]^{-1/2})$ 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 $dtbig \le \min(2/N,$ $\frac{1}{\lvert V_{h\ \max}\rvert}\left[\left(\frac{1}{\Delta x}\right)^2+\left(\frac{1}{\Delta y}\right)^2\right]^{-1/2}, \frac{\Delta z}{w_{\max}})$ 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 ≤ *tacoef* ≤ 1.0. When *tacoef*=0.5, the current and future time levels have equal weight. *tacoef*=0.6 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.

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	Constant 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.

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 $dtsml \le \frac{1}{c_s}\left[\left(\frac{1}{\Delta x}\right)^2+\left(\frac{1}{\Delta y}\right)^2+\left(\frac{1}{\Delta z_{\min}}\right)^2\right]^{-1/2}$ where c_s is the maximum sound wave speed. When *vimplct*=1, the constraint is $dtsml \le \frac{1}{c_s}\left[\left(\frac{1}{\Delta x}\right)^2+\left(\frac{1}{\Delta y}\right)^2\right]^{-1/2}$. 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 turn 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 spactial 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 in side the code.
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 automatically 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 $rbclbc \neq 0$, 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.	$0.0 \leq rlxlbc \leq 0.5$. 0.05 recommended.

bbc	Options for the bottom boundary condition.	1 = rigid lid or rigid ground. 2 = periodic (top and bottom at the same time). 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 (top and bottom at the same time). 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., $nx = 2^p * 3^q * 5^r$, $ny = 2^p * 3^q * 5^r$. This time, 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 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. It is in the *yyyy-mn-dd.hh:mm:ss* format, where *yyyy* , *mn*, *dd*, *hh, mm* and *ss* are integers for 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 in.
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 lightly 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). τ = 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} is Section 8.6.	Similar to the second-order computational mixing coefficient. Although most of the time, extra 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 enters as follows: $(\frac{du}{dt})_{Coriolis} = f\hat{v} - \tilde{f}w$ $(\frac{dv}{dt})_{Coriolis} = -f\hat{u}$ $(\frac{dw}{dt})_{Coriolis} = \tilde{f}u$ 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. 4 = option 2, but f and $\tilde{f}$ are functions of latitude.

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: $(\frac{du}{dt})_{Coriolis} = f\,\hat{v} - \tilde{f}\,w$ $(\frac{dv}{dt})_{Coriolis} = -f\,\hat{u}$ $(\frac{dw}{dt})_{Coriolis} = \tilde{f}\,u$ 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.	*coriotrm* = 2 is used 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. In a sense, the balancing horizontal pressure gradient is introduced through base-state wind.

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). 4 is recommended.

trbisotp	Option for isotropic subgrid scale turbulence.	= 0, the turbulence is assumed to be anisotropic, Use 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, it is reset to 1 in the code. 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 usual chosen so that the e-folding time of damping on the 2Δ waves [$\tau=\Delta^2/(\pi^2\ tmixcst)^{-1}$] is short enough to effectively kill them off. 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.	$0 < kmlimit \leq 1.0$. 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≠0*	Linear stability analysis requires that $cfcm2h\ \Delta t < 1/8\ (\Delta t = dtbig)$. Use a value just large enough to kill off shortest (2Δ) waves. The corresponding e-folding time should be at least 10 *dtbig*. Too large a value 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≠0.*	Linear stability analysis requires that $cfcm2v\ \Delta t < 1/8\ (\Delta t = dtbig)$. Use a value just large enough to kill off shortest (2Δ) waves. The corresponding e-folding time should be at least 10 *dtbig*. Too large a value 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 to 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}/Δ_h^4 , and α_{4H} =*cfcm4h* Δt . Used only when *cmix4th*≠*0*	Linear stability analysis requires that *cfcm4h* Δt < *1/8 (Δt=dtbig).* Use a value just large enough to kill off shortest (2Δ) waves. The corresponding e-folding time should be at least 10 *dtbig*. Too large a value 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}/Δ_v^4 , and α_{4v} =*cfcm4v* Δt . Used only when *cmix4th*≠*0*	Linear stability analysis requires that *cfcm4v* Δt < *1/8 (Δt=dtbig).* Use a value just large enough to kill off shortest (2Δ) waves. The corresponding e-folding time should be at least 10 *dtbig*. Too large a value 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.
!         Default is 1.

!  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 on (use when $\Delta x \sim \Delta z$); = 2, anisotripic divergence damping on (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 $\frac{\partial(\bar{\rho}\vec{V})}{\partial t} \sim divdmpndh\frac{\Delta x \Delta y}{\Delta \tau}\nabla_h D + divdmpndv\frac{(\Delta z)^2}{\Delta \tau}\frac{\partial D}{\partial z}\vec{k}$ 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* ≠ 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* ≠ 0.	A sponge layer depth that is 1/3 or more of the total domain depth is recommended.

Moist Processes (&microphysics)

Parameter	Definition/Purpose	Options/Suggested Values
moist	Option for moist processes.	= 0, completely dry run, all processes relating to moisture are turned off. = 1, moist processes turned on.
mphyopt	Option for microphysics parameterizations.	= 0, microphysics turned off = 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.

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, WRF Betts-Miller-Janjic cumulus parameterization = 5, WRF new Kain-Fritsch scheme (April 2002: KF_ETA)
wcldbs	Vertical motion (m/s) required at cloud base for convection to occur. Used when *cnvctopt = 1* only.	Scale dependent.
confrq	Time interval (s) between cumulus parameterization updates. Used when *cnvctopt ≠0* only.	From 10 minutes to 1 hour depending on problem scale.
idownd	Option to include downdraft in Kuo scheme. Used when *cnvctopt = 1* only.	0 = no downdraft. 1 = a simple downdraft model. 1 recommended.

```
!  kffbfct   Factor for Kain-Fritsch scheme,to feed convectively
!            generated rainwater into grid-resolved rainwater
!            (or snow) field. kffbfct is the fraction of available
!            precipitation to be fed back (0.0 - 1.0).
!            =0.0, no feed back;
!            =1.0, all convective rainwater feed back,
!                  so no cumulus rainfall in this case.
!            0.0 < kffbfct <= 1.0 recommended when horizontal grid
!            spacing is less than 25km.
!
!  kfsubsattrig Turn on sub-saturation in the Kain-Fritsch scheme
!           = 0, off
!           = 1, on
!
!  The following four parameters are used by Kuo scheme only.
!
```

```
!  wcldbs    Vertical motion needed at cloud base for convection.
!  qpfgfrq   Frequency of grid parameters' updates in seconds
!  idownd    Downdraft flag.
!         = 0, no downdrafts;
!         = 1, simple downdraft model.
!
!  impfallopt   Option for vertically implicit fall velocity scheme
!           = 0, explicit scheme
!           = 1, implicit scheme
!  fallvalpha   Weight coefficient for time level n+1 in implicit scheme
!  fallvbeta    Weight coefficient for time level n in implicit scheme
!
!  fallopt   Option for selecting various fall velocity schemes
!           = 1, Lin fall velocity formulation and coeff.
!           = 2, Ferrier (1994) formulation and updated coefficients.
!
!  subsatopt Option for allowing condenstation to occur before
!           100% relative humidity is required.
!           Turned on only for relatively course grid resolutions.
!         = 0, Condensation occurs when RH >=100%
!           1, Condensation occurs when RH >= rhsat, which is
!             a user specified value.
!           2, RH for condensation (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)))
!
!  rhsat       Threshold of RH for condensation
!  rhsatmin    Used when subsatopt=2. Minimum threshold of RH for
!            condensation for a grid size of dx_rhsatmin
!  dx_rhsatmin Used when subsatopt=2. The physical grid distance (m)
!            for condensation to occur when RH=rhsatmin.
!
!            Suggested: 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: dx_rhsat100=5000.
```

Atmospheric Radiation Parameterization (&radiation)

```
!  radopt    Option to switch on/off radiation physics
!         = 0, No radiation physics;
!         = 1, Simplified surface radiation physics;
!         = 2, Atmospheric radiation transfer parameterization.
!         = 3, OASIS data
!
!  Notes: 1) When sfcphy is chosen to 3 or 4, radopt=0 will be reset
!           to 1 in order to compute the surface energy balance for
!           soil model.
!
!  radstgr   Option for radiation computing at staggering points (used by
!           radopt = 2 only).
!
!         = 0, No staggering; Radiation calculation on x-y plane is at
!              all points;
!         = 1, staggering; 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. 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).
!              This scheme can reduce ALMOST HALF of radiation calculation.
!
!                     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
!
!              On boundary, the zero-gradient is assumed.
!
!       NOTE: if nx or ny is even and radstgr=1, the message passing 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.
!
!      NOTE: the MPI bug with odd nx and ny has been fixed.
!           However, when either (nx-3)/nproc_x or (ny-3)/nproc_y is odd,
!           the message passing version will also not produce identical
!           results with one serial processor run.
!
!  rlwopt    Option to choose the longwave schemes.
!         = 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. The
!              computation is faster with this option.
!         = 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
!          0- no topograpghic shade
!          1- topgraphic shade
!          2- topgraphic shade for a idealized valley
!             uniform in the north south direction
!
!  dtrad     Time interval (seconds) to update the radiation forcing
!           (used by radopt = 2 only).
!
!  raddiag   Option to dump radiation variables to a file in GrADS
!           format for diagnostic review. The frequency is controled by
!           dtrad (used by radopt = 2 only).
!
!         = 0, no such dump
!         = 1, dump to a file with a name like 'runname.radout'
!              and its control file has a name like '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  used specified value cdhwtr.
```

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 x 10^{-3} and *cdmwtr* = 1 x 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 x 10^{-3} and *cdmwtr* = 1 x 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* ~ 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 for *pbldopt* = 1.	Case dependent.
tqflxdis	Option for distributing heat and moisture fluxes qudratically in a specified depth *dtqflxdis*. Use only when near surface vertical resolution is high (<50m).	= 0, no distribution; = 1, distribution over *dtqflxdis* = 2, with distribution over a depth according to similarity 1 recommended.

```
!  lsclpbl0  PBL length scale used for tkeopt=3
!          (0.25 recommended by Sun and Chang 1986).
!
!  dtqflxdis Depth of flux distribution for tqflxdis=1, 200 m recommended.
!
!  smthflx   Option to smooth surface fluxes
!          = 0, no smoothing
!          = 1, smoothing
!  numsmth   Number of smooth passes (>=1 if smthflx=1)
```

sfcdiag	Flag controlling the diagnostic calculations and printing in the surface physics package. The output is dumped in GrADS format and is separate from ARPS history file.	= 0, no diagnostics. = 1, with diagnostics.

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

Parameter	Definition/Purpose	Options/Suggested Values
sfcdat	Option for specifying the surface characteristics data for non-restart runs. 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 USDA definition. We added two categories for ice and water. Used when *sfcdat*=1 only.	$1 \leq$ *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.
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 *sfcdtfl**80.	String in quotes. Not to exceed 80 characters in length. *e.g., sfcdtfl='arpssfc.data'*.
soilinit	Option for initializing the time-dependent arrays of the soil model for *initopt* □2. 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*. 3 = same as option 2, except when *initopt*=3 and the initialization file *inifile* contains the data arrays, data in *inifile* will be used instead.

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.
tsoil0	User specified constant initial deep soil temperature (K). Used only when *soilinit* =1.	Problem dependent.
wetsfc0	User specified constant initial equivalent surface soil moisture (ND). Used only when *soilinit* =1.	Problem dependent.
wetdp0	User specified constant initial equivalent deep soil moisture (ND). Used only when *soilinit* =1.	Problem dependent.
wetcanp0	User specified constant initial canopy moisture (ND). Used only when *soilinit* =1.	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'*.
dtsfc	Time step size (*s)* used by the soil model time integration.	*dtsfc* ≤ *dtbig* recommended.

```
!  The following surface parameters are valid for sfcphy = 3 and 4:
!
!
!  styp     Soil type (an integer). Used if sfcdat=1.
!           The soil type is based on USDA definitions along with
!           categories for ice and water.
!
!           01    Sand
!           02    Loamy sand          11 17 23
!           03    Sandy loam          14 20 26 27
!           04    Silt loam
```

```
!         05   Loam               12 18 24
!         06   Sandy clay loam      15 21 28
!         07   Silty clay loam
!         08   Clay loam           13
!         09   Sandy clay          19 25
!         10   Silty clay          16 22
!         11   Clay               29 30 31
!         12   Ice                34
!         13   Water               00
!
!         Note: The numbers on the right hand side above represent
!              Mylne and Henderson-Sellers soil classes.
!
!         Default: 10 for Norman, Oklahoma
!
!  vtyp     Vegetation type (an integer). Used if sfcdat=1.
!
!         01   Desert
!         02   Tundra
!         03   Grassland
!         04   Grassland with shrub cover
!         05   Grassland with tree cover
!         06   Deciduous forest
!         07   Evergreen forest
!         08   Rain forest
!         09   Ice
!         10   Cultivation
!         11   Bog or marsh
!         12   Dwarf shrub
!         13   Semidesert
!         14   Water
!
!         Default: 3 for Norman, Oklahoma
!
!  lai      Leaf Area Index. Used if sfcdat=1. Default: 0.31
!
!  roufns0  Surface roughness. Used if sfcdat=1. Default: 0.01
!
!  veg0     Vegetation fraction. Used if sfcdat=1. Default: 0.3
!
!  sfcdtfl  Data file containing the surface characteristics
!         (soil and vegetation type, leaf area index and surface roughness).
!
!  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)
```

```
!
!  nzsoil   Number of soil layers. Maximum number of levels is 100.
!
!  dzsoil   Averaged vertical grid spacing in transformed
!           computational space (m).
!
!  zrefsoil Reference height of the surface (below ground level) (m).
!
!  tsoilint(1:nzsoil)  Soil temperatures (K)
!  qsoilint(1:nzsoil)  Soil moisture (m**3/m**3)
!
!        If soilinit=1, then every tsoil and qsoil level must be
!                  explicitly defined in the input file below.
!
!  soilstrhopt    Grid soil stretching option.
!          = 0, no vertical stretching;
!          = 1, vertical stretching with f=z**3 function for dz;
!          = 2, vertical stretching with hyperbolic tangent (see User's Guide).
!  soildzmin     Minimum vertical grid spacing in physical space (m). Used
!           if soilstrhopt = 1 or 2.
!
!  soildlayer1   Height (m) of the layer beneath which stretching is not applied.
!           0.0 =< dlayer1 < (nz-3)*dz
!
!  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.
!
!  sfcfmt   Format of data file containing surface characteristics.
!       = 1, Fortran unformatted (default);
!       = 3, HDF4 (uncompressed).
!
!  soilinit Soil model variable initialization option used when initopt.ne.2.
!
!       = 1, Soil model variables are initialized using input parameters;
!
!       = 2, Soil model variables are initialized using values found
!           in file soilinfl.
!           For variables missing in soilinfl, the values in initial
!           file inifile will be used when initopt=3. In another word,
!           the values in soilinfl take precedence over those in inifile.
!
```

```
!       = 3, As soilinit=2, except that the values found in inifile take
!            precedence over those found in soilinfl.
!
!       = 4, Soil temperature variables are initialized by adding
!            offsets to the surface air temperatue, 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
!
!       This option is not used when initopt=2.
!       When initopt=2, data in the restart file will be used.
!
!  soilmodel_forced  Option for forcing radiation data with surface obs.
!       = 0,  Non-forced mode (default)
!       = 1,  Forced mode (Surface data must be available)
!               (Likely used in conjunction with soilinit=5)
!
!
!  nstyp    The number of soil types per grid point.
!
!  ptslnd0  Initial land surface potential temperature (K).
!         Used by option soilinit=1.
!
!  ptswtr0  Initial water surface potential temperature over water (K).
!         Used by option soilinit=1.
!
!  wetcanp0 Initial canopy moisture. Used by option soilinit=1.
!
!  snowdpth0 Initial snow depth (m). Used by option soilinit=1.
!
!  tsprt    Offset of tsfc from sfc air temperature
!
!  t2prt    Offset of tsoil from sfc air temperature
!
!  wgrat    Saturation rate of sfc soil moisture
!
!  w2rat    Saturation rate of deep soil moisture
!
!  soilinfl Data file containing the initial values of soil model variables
!         (ground surface temperature, deep soil temperature,
!          ground surface soil moisture, deep soil moisture, and
```

```
!         canopy moisture)
!
!  soilfmt  Format of data file containing initial values of soil model
!         variables.
!       = 1, Fortran unformatted (default);
!       = 2, OASIS testing.
!       = 3, HDF4 (uncompressed).
!
!  tsoil_offset Option for including seasonal deep and skin layer temperature
!            offset in the two-layer soil model.
!           = 0 Not included
!           = 1 Constant throughout the domain
!           = 2 Spatially dependent offset (not implemented yet)
!
!  tsoil_offset_amplitude The amplitude of the annual cycle of the difference
!           (offset) in deep and skin layer soil seasonal-mean temperatures
!
!  dtsfc    Time step for surface (soil) model integration. dtsfc =< dtbig.
!
!  Note: The above options are effective only when sfcphy = 3 or 4.
```

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.

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 on.	0 = no automatic translation 1 = use cell-tracking algorithm to estimate the new grid translation speed, and try to bring 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.
window	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. Formats 4 and 8 are more compact. ASCII is most portable but the files are very large. HDF and NetCDF are portable if respective libraries are installed. Savi3D and GrADS are visualization/ graphic software packages. See Section 10.1 for detailed descriptions. Note that NetCDF format supported is stopped since version 4.1.5.	0 = no history dump. 1 = unformatted binary (big endian on little endian machines whenever compiler supports it). 2 = ASCII. 3 = NCSA HDF version 4. 4 = packed 16-bit binary (not available on Cray). 5 = Savi3D MeRAF. 6 = binary with grid point skipping in partial domain. 7 = NetCDF. 8 = Packed 16-bit NetCDF. 9 = GrADS. 10 = GRIB 11 = Vid5D 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 ≤ *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.	

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 of maximum size 100, specifying the times when history dumps are created. Used on *hdmpopt*=2.	

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 = 2, 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 dump 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 = no dump. 1 = with dump. When *lbcopt* ≠ 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 dump for the purpose of automatic detection.	= 0, do not create a ready file. = 1, create a ready file;
grdout	Option to dump 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 dump 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 dump 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 dump 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. In this case, ice variable dumping is turned off too.
iceout	Option to dump 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 dump turbulent kinetic energy into the history file.	0 = no. 1 = yes. None zero only when *tmixopt* = 4.

trbout	Option to dump turbulent mixing coefficient array (k_m) into the history file.	0 = no. 1 = dump.
rainout	Option to dump surface accumulated rainfall arrays into the history file.	0 = no. 1 = dump. Set to 0 if the model run is dry.
sfcout	Option to dump soil model time-dependent arrays into the history file.	0 = no. 1 = yes. Set to 0 if surface physics is off.
landout	Option to dump surface characteristics arrays into the history file.	0 = no. 1 = yes.
radout	Option to dump radiation arrays in history dump	0 = no. 1 = yes.
flxout	Option to dump 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 dump; = 1, binary format = 3, (uncompressed) HDF
soildmp	Option to create soil model initial condition format data files.	= 0, no dump; = 1, binary format = 3, (uncompressed) HDF

terndmp	Option to write out an ARPS terrain data file.	= 0, no dump; = 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.	$0 \leq$ *tfmtprt* . Setting *tfmtprt* to zero switches off formatted printing.
tenergy	Time interval between printouts of domain energy statistics (not well 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 the dumps of HDF-image files.	User discretion.
pltopt	Option for generating graphic plots during model run.	Not implemented.
tplot	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
dir_extd	The directory that contains the external data files that are 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='./'*.
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.
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'.*

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

Parameter	Definition/Purpose	Options/Suggested Values
schmopt	Option for the scheme used to generate the distribution of soil and vegetation data set for ARPS.	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 fore-ground region and data from the surface characteristics database in the background. 3 = data from the surface characteristics database in the entire domain.
fgbgni	Beginning index (i) in x-direction of the foreground region.	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.	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.	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.
vtypfl	File name of vegetation class data. Character *vtypfl* *80.	Character string. *e.g, vtypfl='arpssfc.data /owe14d.data*'
ndvifl	File name of NDVI class data. Character *ndvipfl* *80.	Character string. *e.g., ndvifl='arpssfc.data /ndvi.data*'
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.