ARPS 4.0 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. All parameters except the model dimension parameters are specified in a model input file, *arps40.input*. The input file is in NAMELIST format (see the comments in *arps40.input* for an explanation of the 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. The model dimension parameters that define the grid size are set inside *dims.inc*. Whenever *dims.inc* is modified, one should enter the **makearps** command, so that ARPS driver program *arps40.f* will be compiled using the new *dims.inc* file.

In addition to the parameters used for model execution, the NAMELIST input file *arps40.input* also contains a number of extra parameters that are used only by the data preprocessors. Since the preprocessors share certain parameters with ARPS40, *e.g.*, those for grid setup, using the same input file for both the preprocessors and the main model program reduces the chance of errors caused by inconsistent parameter settings.

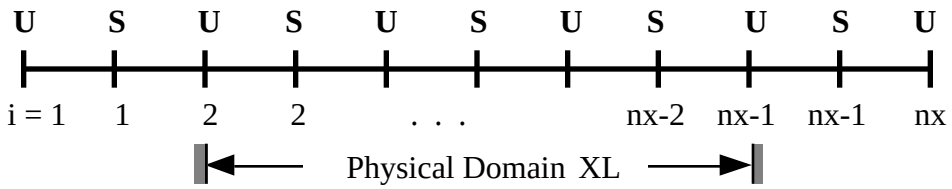
This chapter provides a reference guide on how to choose appropriate values for these control parameters. Recommended values are provided for some parameters.

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. See the Table of Contents for the order in which the groups are listed.

Model Dimension Parameters (in file dims.inc)

Parameter	Definition/Purpose	Options/Suggested Values
nx *(*set in ***dims.inc****)*	Number of grid points in the ξ (or x) direction.	Any value ≥ 4. For 2-D y-z or 1-D single column mode, set nx to 4.

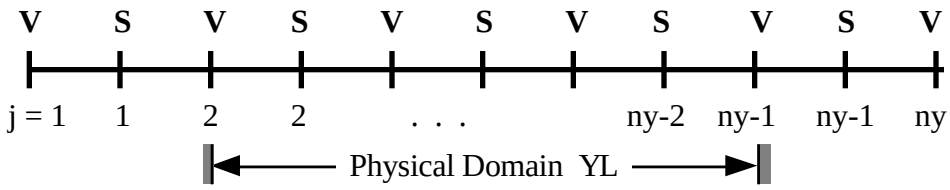
Physical dimension of the model domain in the x (or ξ) direction is $XL = (nx-3)dx$. The grid structure in the x or ξ-direction is:



where S indicates the scalar point and U the u-velocity point.

ny (set in ***dims.inc***)	Number of grid points in the η (or y) direction.	Any value ≥ 4. For 2-D x-z or 1-D single column mode, set ny to 4.

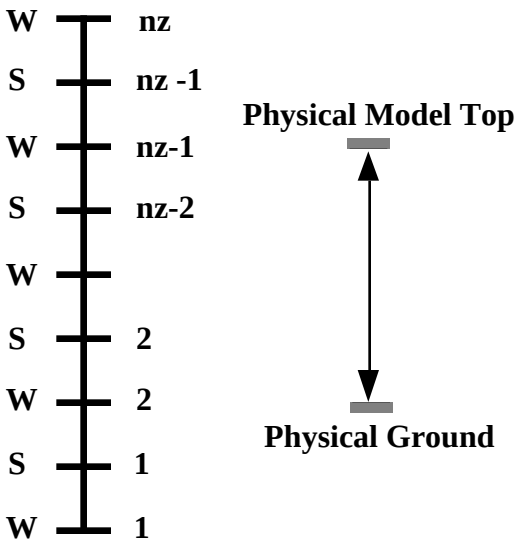
Physical dimension of the model domain in the y (or η) direction is $YL = (ny-3)dy$. The grid structure in the y or η-direction is:



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

nz (set in ***dims.inc***)	Number of grid points in the ζ (or z) direction.	Any value ≥ 4.

Height above ground of the model top boundary is $ZH = (nz-3)dz$. The grid structure in the z or ζ-direction is:



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

[Note that the model defines one more fake point for the scalars and velocity components outside the physical boundaries to facilitate the implementation of boundary conditions.]

Experiment Identification Parameters (&comment_lines)

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'.*

Model Run Name (&runname)

Parameter	Definition/Purpose	Options/Suggested Values
runname	A character string containing the pertinent information for this run. The initial characters before a blank space, a comma or otherwise the first 6 characters will be used to construct history data and restart data dump file names. This string is written into the output data file. Character *runname**80.	Use up to six characters to identify this experiment and data files produced, *e.g., runname='arps40'*.

Model Geometry Configuration Parameters (&model_configuration)

Parameter	Definition/Purpose	Options/Suggested Values
runmod	Option for configuring the model geometry. 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 a single column mode, 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 is run in 3-D. 2 = model is run in 2-D x-z slice mode. 3 = model is run in 2-D y-z slice mode. 4 = model is 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 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 *inidate**19.	Character string in quotes. Choose to match the model time zero, not the time when the model is restarted. *e.g., initime ='1977-05-20.21:00:00'*.

initopt	Option for initializing the time dependent fields.	1 = self-initialization using analytic functions. 2 = initialize using restart data from a previous run. 3 = initialize from an externally-supplied data file.
inibasopt	Option for initializing the base state variables. This initialization is done for all options of *initopt.* For *initopt* = 2 or 3, the base state arrays initialized this way will be overwritten if the restart or 3-D initial data set contains the base-state arrays. In that case, the values read in replace the original values.	1 = initialize the base state using a single sounding. 2 = isentropic atmosphere. 3 = isothermal atmosphere. 4 = atmosphere with a constant static stability. 5 = an analytic thermo-dynamic 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 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 the code to define the profiles. Currently, the model sets *ubar=vbar*=0.0.
ubar0	Constant u-velocity (m/s) for the initial and base-state wind when *inibasopt* ≠ 1 and *viniopt* = 1.	Problem dependent.
vbar0	Constant v-velocity (m/s) for the initial and base-state wind when *inibasopt* ≠ 1 and *viniopt* = 1.	Problem dependent.

pt0opt	Option for specifying the initial potential temperature perturbation field. This option is used only when *initopt*=1.	0 = no 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)*Δζ* (*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*.
buoyopt	Option to turn the buoyancy term on and off. The buoyancy should be turned on except for special cases.	0 = buoyancy is turned off. 1 = buoyancy is turned on.

sndfile	Name of the input sounding file using to initialize the model base state variables. A sounding file has to be present when *initopt*=1. The sounding file format is defined in Section 8.4.1. Character *sndfile**80.	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. Character *rstinf**80.	Character string in quotes. *e.g., rstinf='arps40.rst003600'*.
inifmt	Format flag for the initial data files: *inifile* and *inibgf*. Standard ARPS history data dump formats are used by the initial data files (*inifile* and *inibgf*). Note: not all formats are available for initialization.	1 = unformatted binary. 2 = ASCII. 3 = NCSA HDF. 4 = packed binary. 7 = NetCDF. 8 = Packed NetCDF. Formats 5 and 6 are not available for initializtion.
inifile	Name of the initial data file containing the time dependent arrays, and, possibly, the base-state arrays and grid coordinate arrays depending on the values of flags inside the data. When the base state and grid coordinate arrays are present, *inibgf* will not be read. Used only when *initopt*=3. Character *inifile**80.	Character string in quotes. *e.g., inifile='arps40.bin003600'*.
inibgf	Name of the initial data file containing the base state and grid coordinate arrays. Used only when *initopt*=3. Character *inibgf**80.	Character string in quotes. *e.g., inibgf='arps40.bingrdbas'*.

Terrain Initialization Parameters (&terrain)

Parameter	Definition/Purpose	Options/Suggested Values
ternopt	Model terrain option. When *ternopt* =0, 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 $$h = \frac{h_m}{1 + \left[\frac{x - x_c}{x_r}\right]^2 + \left[\frac{y - y_c}{y_r}\right]^2}$$ 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.
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.
terndta	Name of the terrain data file. Used only by option *ternopt*=2. This data file can be prepared by terrain data preprocessor ARPSTRN. Character *terndta**80	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. 1.0 recommended.
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) of the center of model domain.	Anywhere on the globe.

Map Projection Parameters (&projection)

Parameter	Definition/Purpose	Options/Suggested Values
mapproj	Map projection option. Note: Map factor is included in ARPS 4.0 pre- and post-processors, but it is not yet implemented in ARPS 4.0 dynamic equations. 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 recommended for model runs). A projection that has a map factor close to 1 is recommended.
trulat1 ***trulat2***	1st true latitude (degrees north) of map projection 2nd true latitude (degrees north) of map projection *trulat2* is used only by Lambert projection.	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.

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 non-restart run, *tstart* is usually set to zero. 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 s is recommended.
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 is recommended.
ptsmlstp	Option for integrating internal gravity wave modes (the potential temperature equation) inside small time steps. When doing so, the stability limit for *dtbig* becomes independent of the static stability.	0 = integrate θ equation in large time steps. 1 = integrate θ equation inside small time steps. If advection is the most restrictive process on the big time step size, as is usually the case for storm-scale simulations, 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.
tacoef	Backward-in-time bias coefficient (non-dimensional, ND) used by vertically implicit *w* and *p* solver (when *vimplct* = 1) *tacoef* = β in Eqs. (6.2.31c) and (6.2.31d)	0.5 ≤ *tacoef* ≤ 1.0. When *tacoef*=0.5, there is not biasing. 0.6 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 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. 2 is recommended.
sadvopt	Option for scalar advection.	1 = second-order advection. 2 = fourth-order in horizontal and second-order in vertical advection. 2 is recommended.

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.
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).
tbc ***bbc***	Options for the top and 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.

rbcopt	Option for wave-radiating lateral boundary conditions (RBC).	1 = Klemp and Wilhelmson type radiation condition that uses user-specified constant phase speed (see *c_phase* below). 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 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 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 are 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.
pdetrnd	Option to turn on pressure detrending procedure. (The domain average perturbation Exner function is removed.)	0 = pressure detrending off. 1 = pressure detrending on. 0 recommended.

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, $\hat{u} = u - \bar{u}$ *and* $\hat{v} = v - \bar{v}$.	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. If *coriotrm* = 1 no balancing is imposed. Total *u* and *v* are used in the Coriolis formulation. If *coriotrm* = 2 the Coriolis terms are modified from their standard formulation so that *f(u-ubar)* and *f(v-vbar)* are used in place of *fu* and *fv*. Here *f*ubar* and *f*vbar* represent the geostrophic pressure gradient forces associated with ubar and vbar. This option is not used if *coriopt* = 0.	It may be desirable to impose an approximate initial geostrophic balance (*coriotrm* = 2) if the model is initialized from a single sounding (the base-state pressure gradient being zero).

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 parameterization. 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 = turbulence is anisotropic. 1 = turbulence is isotropic. *trbisotp* = 0 should be used when the grid aspect ratio is large ($dx/dz \geq 100$).
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. $\Delta=\left(\Delta x\Delta y\Delta z\right)^{1/3}$, $\Delta_H=\left(\Delta x\Delta y\right)^{1/2}$ 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 [$T_e=\Delta^2/(\pi^2\ ^t mixcst)^{-1}$] is short enough to effectively kill them off. This background mixing is not necessary when computational mixing is activated.
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.	$0 < kmlimit \le 1.0$. 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 Δt < 1/8 (Δt=dtbig).* Recommended value is 0.001 s^{-1}.
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 Δt < 1/8 (Δt=dtbig).* Recommended value is 0.001 s^{-1}.
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. This option is preferred to the second-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≠0*	Linear stability analysis requires that *cfcm4h* $\Delta t < 1/8$ (Δt=*dtbig*). Recommended value is 0.001 s^{-1}.
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≠0*	Linear stability analysis requires that *cfcm4v* $\Delta t < 1/8$ (Δt=*dtbig*). Recommended value is 0.001 s^{-1}.

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 = damping activated. 1 is recommended.
divdmpnd	Coefficient of divergence damping (ND). The damping term has the form $\frac{\partial(\bar{\rho}\vec{V})}{\partial t} \sim divdmpnd \frac{\Delta^2}{\Delta\tau}\nabla\left[\nabla.(\bar{\rho}\vec{V})\right]$ on the right hand side of the momentum equations, where Δ = min (Δx, Δy, Δz) is the grid scale. $\Delta\tau = dtsml$. Used only when *divdmp≠0*.	*divdmpnd*< 0.75. *divdmpnd* = 0.05 recommended. 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.

Moist Processes (&microphysics)

Parameter	Definition/Purpose	Options/Suggested Values
moist	Option for moist processes.	0 = completely dry run, all code relating to moisture is 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. When *moist*=1 and *mphyopt*=0, only saturation and condensation processes remain.
cnvctopt	Option for convective cumulus parameterization. When cumulus parameterization is turned on, microphysics is turned off.	0 = no cumulus parameterization 1 = Kuo scheme.
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 the Kuo cumulus parameterization updates. Used when *cnvctopt = 1* 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.

Surface Layer Parameterization (&surface_physics)

Parameter	Definition/Purpose	Options/Suggested Values
sfcphy	Option for surface physics (surface fluxes and soil model).	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, surface energy budget and soil model equations are solved.
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.

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 dependent on local time (not yet implemented). 3 = PBL depth is predicted (not yet implemented).
pbldpth0	Specified PBL depth for *pbldopt* = 1.	From 500 m to 2 km. Should be larger than zero when surface flux distribution is on (see below).
sflxdis	Option for linearly distributing surface fluxes in the PBL depth.	0 = no distribution, all fluxes are applied at the ground surface. 1 = all fluxes are distributed. 2 = only heat and moisture fluxes are distributed. 2 recommended.
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 = diagnostics.

Soil Model and Surface Energy Budget (&surface_ebm)

Parameter	Definition/Purpose	Options/Suggested Values
sfcdat	Option for specifying the surface characteristics data 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 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 ≤ *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 ≤ *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*= '*arps40.soilinit*'.
dtsfc	Time step size (*s)* used by the soil model time integration.	*dtsfc* ≤ *dtbig* 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.

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 (&output)

Parameter	Definition/Purpose	Options/Suggested Values
dirname	Name of the directory into which output data are written. Character *dirname**80. If the name of the current directory is desired, set *dirname* ='./'.	Character string in quotes. Not to exceed 80 characters in length. Default ='./' for current directory.
tfmtprt	Time interval (s) between formatted variable printouts in standard output file.	0 ≤ *tfmtprt* . Setting *tfmtprt* to zero switches off formatted printing.
thisdmp	Model time interval (s) between history data file writing.	0 ≤ *thisdmp* . Setting *thisdmp* to zero switches off history dump file writing. It is reset to the nearest integer multiple of d*tbig.*
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.	0 = no history dump. 1 = unformatted binary (IEEE binary on Cray). 2 = ASCII. 3 = NCSA HDF. 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 (not yet available).
exbcdmp	Option to write files that can be used as external boundary condition data at the same time as the history data files. The data will not be written if the history data dump is turned off.	0 = no dump. 1 = dump.

exdtadmp	Option to dump out the files that contain the external data arrays interpolated to the current model time. These files are in the standard history dump format and will be generated at the same time as the history data files.	0 = no dump. 1 = 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 cut the disk usage by as much as a half.
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 dump. 1 = dump. 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 dump. 1 = dump. 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 dump. 1 = dump. 1 recommended.
mstout	Option to dump moisture variable arrays (q_v, q_c and q_r) into the history file.	0 = no dump. 1 = dump. 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 dump. 1 = dump. Set to 0 if the model run is dry.
trbout	Option to dump turbulent mixing coefficient array (k_m) into the history file.	0 = no dump. 1 = dump.

rainout	Option to dump surface accumulated rainfall arrays into the history file.	0 = no dump. 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 dump. 1 = dump. Set to 0 if surface physics is off.
landout	Option to dump surface characteristics arrays into the history file.	0 = no dump. 1 = dump.
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.	User discretion.
tenergy	Time interval between printouts of domain energy statistics.	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).	User discretion.

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.

Control Parameters for External Boundary Condition (&exbcpara)

Parameter	Definition/Purpose	Options/Suggested Values
nxebc *(*set in ***exbc.inc****)*	The first dimension of external boundary condition arrays (for *lbcopt = 2).*	When *lbcopt* = 1, set *nxebc*=1. When *lbcopt* = 2, set *nxebc=nx.*
nyebc *(*set in ***exbc.inc****)*	The second dimension of external boundary condition arrays (for *lbcopt = 2).*	When *lbcopt* = 1, set *nyebc*=1. When *lbcopt* = 2, set *nyebc=ny.*
nzebc *(*set in ***exbc.inc****)*	The third dimension of external boundary condition arrays (for *lbcopt = 2).*	When *lbcopt* = 1, set *nzebc* = 1. When *lbcopt* = 2, set *nzebc =nz.*
nbrlx	The number of grid points inside the boundary relaxation zone for the Davies type externally forced boundary condition (*lbcopt = 2). nbrlx* $= n_b$ in Eq. (8.6.2).	6 points recommended. A larger value can be used if the domain is large (>50 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)	$0 < brlxhw \leq nbrlx$. 2.3 recommended.

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 *exbcnam**80.	String in quotes. Not to exceed 80 characters in length. *e.g., exbcnam = 'arpsexbc'*.
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 *tinitebd**19.	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.
cbcdmp	The maximum coefficient (s^{-1}) of Rayleigh type damping in the boundary relaxation zone. *cbcdmp* = K_{b0} in Eq. (8.6.2).	Problem dependent. 1/300 s^{-1} recommended for most runs.
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.	Slightly scale dependent. 0.001 s^{-1} recommended.

Parameters Used By Program EXT2ARPS (Not by ARPS40) 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 (Not by ARPS40) 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.