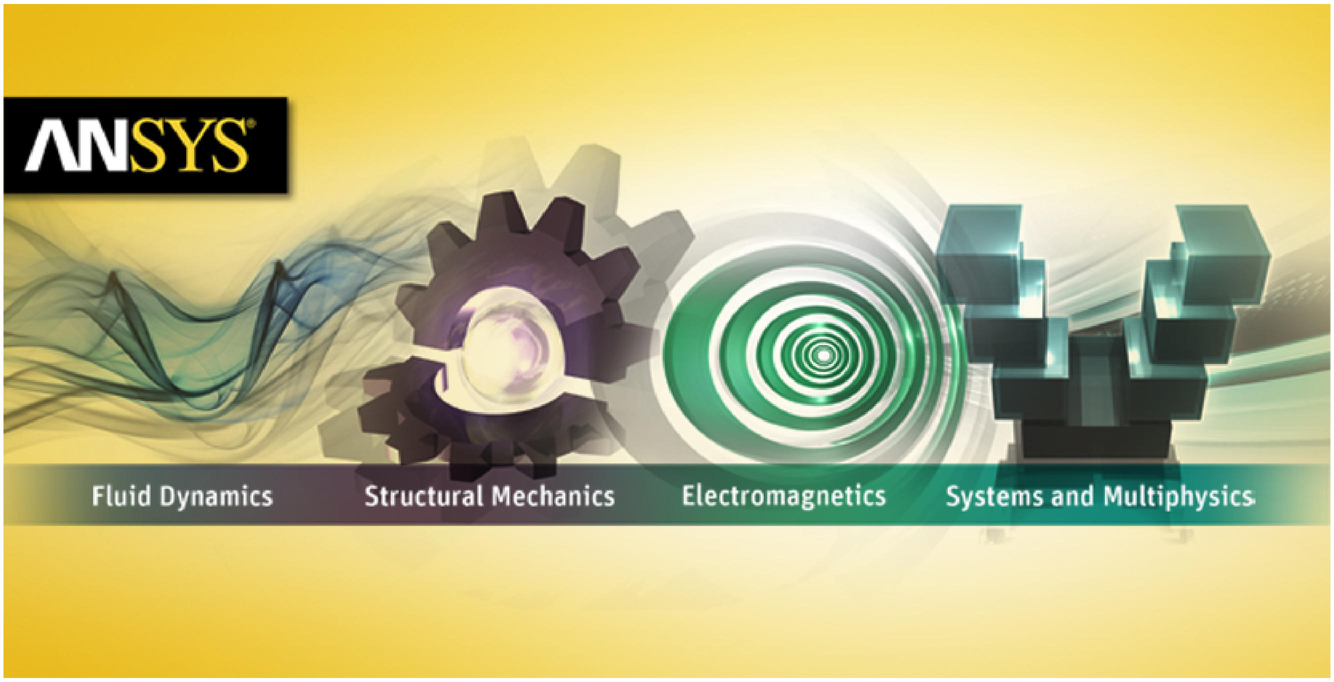

ANSYS FLUENT Text Command List

ANSYS, Inc.
Southpointe
275 Technology Drive
Canonsburg, PA 15317
ansysinfo@ansys.com
http://www.ansys.com
(T) 724-746-3304
(F) 724-514-9494

Release 14.0
November 2011

ANSYS, Inc. is certified to ISO 9001:2008.

Published in the U.S.A.

Table of Contents

Chapter 1: adapt/

Important

Text User Interface commands that take single or multiple zone names support the use of wildcards. For example, to adapt boundary cells (`adapt-boundary-cells`) based on a list of face zone names, use one or more `*` in the name of the zone(s).

`adapt-boundary-cells`
Adapt boundary cells based on a list of face zones.

`adapt-to-gradients`
Adapt mesh based on the gradient adaption function from the selected scalar quantity, the adaption threshold values, and the adaption limits.

`adapt-to-ref-lev`
Adapt cells based on refinement level differences.

`adapt-to-register`

Adapt mesh based on the selected adaption register and adaption limits.

`adapt-to-vol-change`
Adapt cells with large changes in cell volume.

`adapt-to-volume`
Adapt cells that are larger than a prescribed volume.

`adapt-to-y+`
Adapt cells associated with all wall zones based on the specified threshold values and adaption limits.

`adapt-to-y+-zones`
Adapt cells associated with specified wall zones based on the specified threshold values and adaption limits.

`anisotropic-adaption`
Anisotropically refine boundary layers. Cells will be split in the normal direction to the boundary face.

`adapt-to-y*`
Adapt cells associated with all wall zones based on the specified threshold values and adaption limits.

`adapt-to-y*-zones`
Adapt cells associated with specified wall zones based on the specified threshold values and adaption limits.

`change-register-type`
Toggle specified register between refinement and mask.

`combine-registers`
Combine the selected adaption and/or mask registers to create hybrid adaption functions.

`delete-register`
Delete an adaption register.

display-register
Display the cells marked for adaption in the specified adaption register.

exchange-marks
Exchange the refinement and coarsening marks of the specified adaption register.

fill-crsn-register
Mark all cells to coarsen that are not marked for refinement in the adaption register.

free-parents
Delete the hanging node face and cell hierarchy.

free-registers
Delete all adaption and mask registers.

invert-mask
Change all the active cells to inactive cells in a mask register.

limit-register
Apply the adaption volume limit to the selected register.

list-registers
Print a list of the current registers including the ID, description (name), number of cells marked for refinement and coarsening, and the type.

mark-boundary-cells
Mark boundary cells based on a list of zones for refinement.

mark-boundary-normal
Mark cells for refinement based on target boundary normal distance.

mark-boundary-vol
Mark cells for refinement based on target boundary volume.

mark-inout-circle
Mark cells with centroids inside/outside the circular region defined by text or mouse input.

mark-inout-cylinder

Mark cells with centroids inside/outside the arbitrarily oriented cylindrical region defined by text or mouse input.

mark-inout-hexahedron
Mark cells with centroids inside/outside the hexahedral region defined by text or mouse input.

mark-inout-iso-range
Mark cells for refinement that have values inside/outside the specified isovalue ranges of the selected field variable.

mark-inout-rectangle
Mark cells with centroids inside/outside the rectangular region defined by text or mouse input.

mark-inout-sphere
Marks cells with centroids inside/outside the spherical region defined by text or mouse input.

mark-percent-of-ncells
Mark percent of total cell count for adaption based on gradient or isovalue.

mark-with-gradients
Mark cells for adaption based on flow gradients for refinement.

mark-with-ref-lev
Mark cells based on refinement level differences.

mark-with-vol-change
Mark cells with large changes in cell volume for refinement.

mark-with-volume
Mark cells for adaption based on maximum allowed volume.

mark-with-y+
Mark cells associated with all wall zones for refinement or coarsening based on the specified threshold values.

mark-with-y+-zones
Mark only cells associated with specified wall zones for refinement or coarsening based on the specified threshold values.

mark-with-y*
Mark cells associated with all wall zones for refinement or coarsening based on the specified threshold values.

mark-with-y*-zones
Mark only cells associated with specified wall zones for refinement or coarsening based on the specified threshold values.

set/
Enter the adaption set menu.

cell-zones
Set cell zones to be used for marking adaption.

coarsen-mesh?
Turn on/off ability to coarsen mesh.

display-crsn-settings
Prompt for coarsening wireframe visibility and shading, and the marker visibility, color, size and symbol.

display-node-flags
Display color coded markers at the nodes specifying the node type.

display-refn-settings
Prompt for refinement wireframe visibility and shading, and the marker visibility, color, size and symbol.

grad-vol-weight
Control the volume weighting for the gradient adaption function.

init-node-flags
Initialize the node flags.

max-level-refine
Set maximum level of refine in the mesh.

max-number-cells
Limit the total number of cells produced by refinement.

min-cell-volume
Restrict the size of the cells considered for refinement.

min-number-cells
Set limit on the number of cells in the mesh.

reconstruct-geometry
Enable/disable geometry-based adaption.

`refine-mesh?`
Turn on/off mesh adaption by point addition.

`set-geometry-controls`
Set geometry controls for wall zones.

`smooth-mesh`
Smooth the mesh using the quality-based, Laplacian, or skewness methods.

`swap-mesh-faces`
Swap the faces of cells that do not meet the Delaunay circle test.

Chapter 2: define/

`boundary-conditions/`
Enter the boundary conditions menu.

Important

Text User Interface commands that take single or multiple zone names support the use of wildcards. For example, to copy boundary conditions (`copy-bc`) to all zones of a certain type, use a `*` in the name of the zone to which you want to copy the conditions.

`axis`
Set boundary conditions for a zone of this type.

`bc-settings/`
Enter the boundary conditions settings menu.

`mass-flow`
Select method for setting the mass flow rate.

`pressure-outlet`
Select pressure specification method on pressure-outlet boundaries.

`phase-shift/`
Enter the phase shift settings menu.

`multi-disturbances`
Set basic phase-shift parameters.

`extra-settings`
Set other phase-shift parameters.

`copy-bc`
Copy boundary conditions to other zones.

`exhaust-fan`
Set boundary conditions for a zone of this type.

`fan`
Set boundary conditions for a zone of this type.

`fluid`
Set boundary conditions for a zone of this type.

`inlet-vent`
Set boundary conditions for a zone of this type.

`intake-fan`
Set boundary conditions for a zone of this type.

`interface`
Set boundary conditions for a zone of this type.

interior
Set boundary conditions for a zone of this type.

list-zones
Print out the types and IDs of all zones in the console window. You can use your mouse to check a zone ID, following the instructions listed under **Zone** in the **Boundary Conditions Task Page** section of the User's Guide.

mass-flow-inlet
Set boundary conditions for a zone of this type.

modify-zones/
Enter the modify zones menu.

activate-cell-zone
Activate cell thread.

append-mesh
Append new mesh.

append-mesh-data
Append new mesh with data.

copy-move-cell-zone
Create a copy of a cell zone that is offset from the original either by a translational distance or a rotational angle. In the copied zone, the bounding face zones are all converted to walls, any existing cell data is initialized to a constant value, and non-conformal interfaces and dynamic zones are not copied; otherwise, the model settings are the same as in the original zone. Note that if you want the copied zone to be connected to existing zones, you must either fuse the boundaries (see Fusing Face Zones in the User's Guide) or set up a non-conformal interface (Using a Non-Conformal Mesh in ANSYS FLUENT in the User's Guide).

copy-mrf-to-mesh-motion
Copy motion variable values for origin, axis, and velocities from Frame Motion to Mesh Motion.

copy-mesh-to-mrf-motion
Copy motion variable values for origin, axis, and velocities from Mesh Motion to Frame Motion.

create-all-shell-threads
Mark all finite thickness walls for shell creation. Shell zones will be created at the start of the iterations.

deactivate-cell-zone
Deactivate cell thread.

delete-all-shells
Delete all shell zones and switch off shell conduction on all the walls. These zones can be recreated using the command `recreate-all-shells`.

delete-cell-zone
Delete a cell thread.

extrude-face-zone-delta
Extrude a face thread a specified distance based on a list of deltas.

extrude-face-zone-para
Extrude a face thread a specified distance based on a distance and a list of parametric locations between 0 and 1, e.g., 0 0.2 0.4 0.8 1.0.

fuse-face-zones
Attempt to fuse zones by removing duplicate faces and nodes.

list-zones
List zone IDs, types, kinds, and names.

make-periodic
Attempt to establish periodic/shadow face zone connectivity.

matching-tolerance
Set normalized tolerance used for finding coincident nodes.

merge-zones
Merge zones of same type and condition into one.

mrf-to-sliding-mesh
Change the motion specification from MRF to moving mesh.

orient-face-zone
Orient the face zone.

recreate-all-shells
Recreate shells on all the walls which were deleted using the command `delete-all-shells`.

replace-zone
Replace cell zone.

sep-cell-zone-mark
Separate cell zone based on cell marking.

sep-cell-zone-region
Separate cell zone based on contiguous regions.

sep-face-zone-angle
Separate face zone based on significant angle.

sep-face-zone-face
Separate each face in zone into unique zone.

sep-face-zone-mark
Separate face zone based on cell marking.

sep-face-zone-region
Separate face zone based on contiguous regions.

slit-periodic
Slit periodic zone into two symmetry zones.

slit-face-zone
Slit two-sided wall into two connected wall zones.

zone-name
Give a zone a new name.

zone-type
Set a zone's type. You will be prompted for the ID of the zone to be changed and the new boundary type for that zone.

non-reflecting-bc/
Enter the non-reflecting boundary condition menu.

general-nrbc/
Setting for general non-reflecting b.c.

set/
Enter the setup menu for general non-reflecting b.c.'s.

`sigma`
Set NRBC sigma factor (default value 0.15).

`sigma2`
Set NRBC sigma2 factor (default value 5.0).

`turbo-specific-nrbc/`
Enter the turbo specific nrbc menu.

`enable?`
Enable/disable non-reflecting b.c.'s.

`initialize`
Initialize non-reflecting b.c.'s.

`set/`
Enter the set menu for non-reflecting b.c. parameters.

`discretization`
Enable use of higher-order reconstruction at boundaries if available.

`under-relaxation`
Set non-reflecting b.c. under-relaxation factor.

`verbosity`
Set non-reflecting b.c. verbosity level. 0 : silent, 1 : basic information (default), 2 : detailed information for debugging.

`show-status`
Show current status of non-reflecting b.c.'s.

`open-channel-wave-settings`
Open channel wave input analysis.

`openchannel-threads`
List open channel group IDs, names, types and variables.

`outflow`
Set boundary conditions for a zone of this type.

`outlet-vent`
Set boundary conditions for a zone of this type.

`periodic`
Set boundary conditions for a zone of this type.

`porous-jump`
Set boundary conditions for a zone of this type.

`pressure-far-field`
Set boundary conditions for a zone of this type.

`pressure-inlet`
Set boundary conditions for a zone of this type.

`pressure-outlet`
Set boundary conditions for a zone of this type.

`radiator`
Set boundary conditions for a zone of this type. Not supported in ANSYS FLUENT

`rans-les-interface`
Set boundary conditions for a zone of this type.

shadow
Set boundary conditions for a zone of this type.

solid
Set boundary conditions for a zone of this type.

symmetry
Set boundary conditions for a zone of this type.

target-mass-flow-rate-settings/
Enter the targeted mass flow rate settings menu.

set under-relaxation-factor
The default setting is 0.05.

enable targeted mass flow rate verbosity?
Enable/disable verbosity when using targeted mass flow rate. When enabled, it prints to the console window the required mass flow rate, computed mass flow rate, mean pressure, the new pressure imposed on the outlet, and the change in pressure in SI units.

velocity-inlet
Set boundary conditions for a zone of this type.

wall
Set boundary conditions for a zone of this type.

zone-name
Give a zone a new name.

zone-type
Set a zone's type.

custom-field-functions/
Enter the custom field functions menu.

define
Define a custom field function.

delete
Delete a custom field function.

example-cff-definitions
List example custom field functions.

list-valid-cell-function-names
List the names of cell functions that can be used in a custom field function.

load
Load a custom field function.

save
Save a custom field function.

dynamic-mesh/
Enter the dynamic mesh menu.

actions/
Enter the dynamic mesh action menu, where you can initiate manual remeshing (i.e., remeshing without running a calculation).

remesh-cell-zone
Manually remesh a cell zone with option to remesh adjacent dynamic face zones.

remesh-cell-zone-cutcell
Manually remesh a cell zone using the CutCell zone remeshing method, in order to generate a predominantly Cartesian mesh.

controls/
Enter the dynamic mesh controls menu. This text command is only available when the `define/dynamic-mesh/dynamic-mesh?` text command is enabled.

implicit-update-parameters/
Enter the dynamic mesh implicit update menu. This text command is only available when you enable implicit mesh updating using the prompts of the `define/dynamic-mesh/dynamic-mesh?` text command.

motion-relaxation
Specify a value (within the range of 0 to 1) for the motion relaxation, which is applied during the implicit mesh update.

residual-criteria
Specify the relative residual threshold that is used to check the motion convergence during the implicit mesh update.

update-interval
Specify the update interval (i.e., the frequency in iterations) at which the mesh is updated within a time step.

in-cylinder-output?
Enable/disable in-cylinder output.

in-cylinder-parameter/
Enter the dynamic mesh in-cylinder menu.

crank-angle-step
Specify crank angle step size.

crank-period
Specify the crank period.

max-crank-angle-step
Specify maximum crank angle step size.

minimum-lift
Specify minimum lift for in-cylinder valves.

modify-lift
Modify lift curve (shift or scale).

piston-data
Specify the crank radius and connecting rod length.

piston-stroke-cutoff
Specify the cut off point for in-cylinder piston.

position-starting-mesh
Move mesh from top dead center to starting crank angle.

print-plot-lift
Print or plot valve lift curve.

layering?
Enable/disable dynamic-layering in quad/hex cell zones.

layering-parameters/
Enter the dynamic mesh layering menu.

`collapse-factor`
Set the factor determining when to collapse dynamic layers.

`constant-height?`
Enable/disable layering based on constant height, else layering based on constant ratio.

`split-factor`
Set the factor determining when to split dynamic layers.

`remeshing?`
Enable/disable local remeshing in tri/tet and mixed cell zones.

`remeshing-parameters/`
Enter the dynamic mesh remeshing menu to set parameters for all remeshing methods except the Cutcell zone remeshing method.

`cell-skew-max`
Set the cell skewness threshold above which cells will be remeshed.

`face-skew-max`
Set the face skewness threshold above which faces will be remeshed.

`length-max`
Set the length threshold above which cells will be remeshed.

`length-min`
Set the length threshold below which cells will be remeshed.

`must-improve-skewness?`
Enable/disable cavity replacement only if remeshing improves the skewness.

`remeshing-after-moving?`
Enable a second round of remeshing based on the skewness parameters after the boundary has moved.

`remeshing-methods`
Enable/disable remeshing methods.

`size-remesh-interval`
Set the interval (in time steps) when remeshing based on size is done.

`sizing-funct-defaults`
Set sizing function defaults.

`sizing-funct-rate`
Determine how far from the boundary the increase/decrease happens.

`sizing-funct-resolution`
Set the sizing function resolution with respect to shortest boundary.

`sizing-funct-variation`
Set the maximum sizing function increase/decrease in the interior.

`sizing-function?`
Enable/disable sizing function to control size based remeshing.

`six-dof-parameters/`
Enter the dynamic mesh six-dof menu.

`motion-history?`
Enable/disable writing position/orientation of six DOF zones to file.

`x-component of gravity`
Specify x-component of gravity.

y-component of gravity
Specify y-component of gravity.

z-component of gravity
Specify z-component of gravity.

smoothing?
Enable/disable smoothing in cell zones.

smoothing-parameters/
Enter the dynamic mesh smoothing menu.

bnd-node-relaxation
Set the spring boundary node relaxation factor.

bnd-stiffness-factor
Set the stiffness factor for springs connected to boundary nodes.

boundary-distance-method
Set the method used to evaluate the boundary distance for the diffusion coefficient calculation, when diffusion-based smoothing is enabled.

constant-factor
Set the spring constant relaxation factor.

convergence-tolerance
Set the convergence tolerance for spring-based solver.

diffusion-coeff-function
Specify whether the diffusion coefficient for diffusion-based smoothing is based on the boundary distance or the cell volume.

diffusion-coeff-parameter
Set the diffusion coefficient parameter used for diffusion-based smoothing.

max-iter
Set the maximum number of iterations for spring-based solver.

relative-convergence-tolerance
Set the relative residual convergence tolerance for Diffusion smoothing.

skew-smooth-cell-skew-max
Set the skewness threshold, above which cells will be smoothed using the skewness method.

skew-smooth-face-skew-max
Set the skewness threshold, above which faces will be smoothed using the skewness method.

skew-smooth-niter
Set the number of skewness-based smoothing cycles.

smoothing-method
Specify whether the smoothing is spring based or diffusion based.

spring-on-all-shapes?
Enable/disable spring-based smoothing for all cell shapes.

spring-on-deformable-shapes?
Enable/disable spring-based smoothing for triangular / tetrahedral cells in mixed element zones.

dynamic-mesh?
Enable/disable the dynamic mesh solver.

events/

Enter the dynamic mesh events menu.

export-event-file

Export dynamic mesh events to file.

import-event-file

Import dynamic mesh event file.

steady-pseudo-time-control

Enable/disable the pseudo time step control in the graphical user interface.

zones/

Enter the dynamic mesh zones menu.

create

Create or edit a dynamic zone.

delete

Delete a dynamic zone.

insert-boundary-layer

Insert a new cell zone.

insert-interior-layer

Insert a new layer cell zone at a specified location.

list

List the dynamic zones.

remove-boundary-layer

Remove a cell zone.

remove-interior-layer

Remove an interior layer cell zone.

enable-mesh-morpher-optimizer?

Enable the mesh morpher/optimizer. When the mesh morpher/optimizer is enabled, the `define/mesh-morpher-optimizer` text command becomes available.

injections/

Enter the injections menu.

For a description of the items in this menu, see `define/models/dpm/injections`.

materials/

Enter the materials menu.

change-create

Change the properties of a locally-stored material or create a new material.

copy

Copy a material from the database.

copy-by-formula

Copy a material from the database by formula.

data-base/

Enter the material database menu.

database-type

Set the database type.

edit
Edit material.

list-materials
List all materials in the database.

list-properties
List the properties of a material in the database.

new
Define new material.

save
Save user-defined database.

delete
Delete a material from local storage.

list-materials
List all locally-stored materials.

list-properties
List the properties of a locally-stored material.

mesh-interfaces/
Enter the mesh-interfaces menu.

create
Create a mesh interface.

delete
Delete a mesh interface.

draw
Draw specified sliding interface zone.

enforce-continuity-after-bc?
Enable/disable continuity across the boundary condition interface for contour plots in postprocessing.

list
List all mesh interfaces.

make-periodic
Make interface zones periodic.

reset
Delete all sliding-interfaces.

smallest-polygon-size
Set the smallest virtual polygon size.

use-virtual-polygon-approach
Use new virtual polygon approach for interfaces.

Important

Note that case files created after ANSYS FLUENT 6.1 will not show the virtual-polygon option, since it is the default.

mesh-morpher-optimizer/

Enter the mesh morpher/optimizer menu in order to deform the mesh as part of a shape optimization problem. This text command is only available when the `define/enable-mesh-morpher-optimizer?` text command has been enabled.

deformation-settings/

Enter the deformation menu. This text command is only available if you have created a deformation region using the `define/mesh-morpher-optimizer/region/create` text command.

check-mesh

Display a mesh check report in the console for the mesh displayed in the graphics window. The mesh check report provides volume statistics, mesh topology and periodic boundary information, verification of simplex counters, and verification of node position with reference to the x axis for axisymmetric cases. This text command is only available if the `define/mesh-morpher-optimizer/optimizer?` text command is disabled.

deform-mesh

Modify the mesh and update the mesh display in the graphics window based on the parameter and deformation settings. This text command is only available if the `define/mesh-morpher-optimizer/optimizer?` text command is disabled.

reset-all-deformations

Undo any deformations made to the mesh and update the mesh display in the graphics window.

set-constraints

Define the constraints on the boundary zones, in order to limit the freedom of particular zones that fall within the deformation region(s) during the morphing of the mesh.

set-parameters

Define deformation parameters assigned to control points.

morpher-summary

Display a summary of the mesh morpher/optimizer settings in the console. This text command is only available if you have created a deformation region using the `define/mesh-morpher-optimizer/region/create` text command.

optimizer-parameters/

Enter the optimizer menu. This text command is only available when you have created a deformation region using the `define/mesh-morpher-optimizer/region/create` text command and have enabled the `define/mesh-morpher-optimizer/optimizer?` text command.

convergence-criteria

Define the convergence criteria for the optimizer.

custom-calculator

Enter the custom calculator menu, in order to define the objective function as a function of output parameters.

define

Define the custom objective function that will be minimized by the optimizer.

delete

Delete the saved custom objective function.

example-obj-fn-definitions

Print examples of custom objective function definitions in the console.

list-output-parameters

Print a list of the output parameters that can be used to define the custom objective function.

end-commands
Specify the commands (text commands or command macros) that will be executed after the solution has run and converged for a design stage.

initial-commands
Specify the commands (text commands or command macros) that will be executed after the design has been modified, but before ANSYS FLUENT has started to run the calculation for that design stage.

initialize?
Specify that the solution variables should be initialized to the values defined in the **Solution Initialization** task page after deformation, rather than allowing the solution variables to remain the values obtained in the previous design iteration.

iterations-per-design
Define the maximum number of iterations ANSYS FLUENT will perform for each design change.

maximum-designs
Define the maximum number of design stages the optimizer will undergo to reach the specified objective function.

monitor/
Enter the monitor menu in order to plot and/or record optimization history data, i.e., how the value of the objective function varies with each design stage produced by the mesh morpher/optimizer.

clear-opt-hist
Discard the optimization history data, including the associated files.

plot-hist
Display an XY plot of the optimization history data generated during the last calculation. Note that no plot will be displayed if the data was discarded using the `define/mesh-morpher-optimizer/optimizer-parameters/monitor/clear-optimization-monitor-data` text command.

plot?
Enable the plotting of the optimization history data in the graphics window.

write?
Enable the saving of the optimization history data to a file.

objective-function-definition
Specify whether the format of the objective function is a user-defined function, a Scheme source file, or a custom function based on output parameters. The custom function is defined using the text commands in the `define/mesh-morpher-optimizer/optimizer-parameters/custom-calculator` menu.

optimize
Initiate the optimization process. This text command is only available if the `define/mesh-morpher-optimizer/optimizer?` text command is enabled.

optimizer-type
Specify which optimizer is used. For information about how the optimizers function, see "Modeling Flows Using the Mesh Morpher/Optimizer" in the User's Guide.

optimizer?
Enable the use of an in-built optimizer. This text command is only available if you have created a deformation region using the `define/mesh-morpher-optimizer/region/create` text command.

`parameter-settings/`
Enter the parameters menu.

`number-of-parameters`
Define the number of parameters available to be assigned to control points.

`parameter-value`
Define the magnitude of deformation for a parameter. The value you define will be multiplied by the scaling factors defined for the deformation settings. This text command is only available if the `define/mesh-morpher-optimizer/optimizer?` text command is disabled.

`region/`
Enter the region menu in order to define the regions of the domain where the mesh will be deformed in order to optimize the shape.

`create`
Create a new deformation region. The region will be a "box", that is, a rectangle for 2D cases or a rectangular hexahedron for 3D cases. After you have created a deformation region, additional menus will be available in the `define/mesh-morpher-optimizer` menu.

`delete`
Delete a deformation region.

`mixing-planes/`
Enter the mixing planes menu.

`create`
Create a mixing plane.

`delete`
Delete a mixing plane.

`list`
List defined mixing plane(s).

`set/`
Set global parameters relevant to mixing planes.

`averaging-method`
Set the mixing plane profile averaging method.

`under-relaxation`
Set mixing plane under-relaxation factor.

`fix-pressure-level`
Set fixed pressure level using value based on `define/reference-pressure-location`.

`conserve-swirl/`
Enter the menu to set swirl conservation in mixing plane menu.

`enable?`
Enable/disable swirl conservation in mixing plane.

`verbosity?`
Enable/disable verbosity in swirl conservation calculations.

`report-swirl-integration`
Report swirl integration (Torque) on inflow and outflow zones.

`conserve-total-enthalpy/`
Enter the menu to set total enthalpy conservation in mixing plane menu.

enable?
Enable/disable total enthalpy conservation in mixing plane.

verbosity?
Enable/disable verbosity in total-enthalpy conservation calculations.

models/
Enter the models menu to configure the solver.

acoustics/
Enter the acoustics menu.

auto-prune
Enable/disable auto prune of the receiver signal(s) during read- and-compute.

broad-band-noise?
Enable/disable the broadband noise model.

convective-effects?
Enable/disable the convective effects option.

compute-write
Compute sound pressure.

cylindrical-export?
Enable/disable the export of data in cylindrical coordinates.

display-flow-time?
Enable/disable the display of flow time during read-and- compute.

export-source-data-cgns?
Enable/disable the export of acoustic source data in CGNS format.

export-volumetric-sources?
Enable/disable the export of fluid zones.

export-volumetric-sources-cgns?
Enable/disable the export of fluid zones.

ffowcs-williams?
Enable/disable the Ffowcs-Williams-and-Hawkings model.

moving-receiver?
Enable/disable the moving receiver option.

off?
Enable/disable the acoustics model.

read-compute-write
Read acoustic source data files and compute sound pressure.

receivers
Set acoustic receivers.

sources
Set acoustic sources.

write-acoustic-signals
Write on-the-fly sound pressure.

write-centroid-info
Write centroid info.

addon-module
Load addon module.

axisymmetric?
Specify whether or not the domain is axisymmetric.

crevice-model?
Enable/disable the crevice model.

crevice-model-controls/
Enter the crevice model controls menu.

dpm/
Enter dispersed phase model menu.

clear-particles-from-domain
Remove/keep all particles currently in the domain.

collisions/
Enter the DEM collisions menu.

collision-mesh
Input for the collision mesh.

collision-pair-settings/
Supply settings for collisions to a pair of collision partners. You will be prompted to specify the `Impact collision partner` and the `Target collision partner`.

contact-force-normal
Sets the normal contact force law for this pair of collision partners.

contact-force-tangential
Sets the tangential contact force law for this pair of collision partners.

list-pair-settings
Lists the current settings for this pair of collision partners.

collision-partners/
Manage collision partners.

copy
Copy a collision partner.

create
Create a collision partner.

delete
Delete a collision partner.

list
Lists all known collision partners.

rename
Rename a collision partner.

dem-collisions?
Enable/disable the DEM collision model.

list-all-pair-settings
For each pair of collision partners, lists the collision laws and their parameters.

max-particle-velocity
Set the maximum particle velocity that may arise from collisions.

injections/
Enter the injections menu.

create-injection
Create an injection.

delete-injection
Delete an injection.

list-particles
List particle streams in an injection.

modify-all-injections
Enter the menu to set properties for all injections.

injection-type
Define injection type.

number-of-tries
Set the number of stochastic tries.

random-eddy-lifetime?
Turn enable/disable a random eddy lifetime.

stochastic-tracking?
Turn enable/disable stochastic tracking.

time-scale-constant
Set the time scale constant.

rename-injection
Rename an injection.

set-injection-properties
Set injection properties.

interaction/
Set parameters for coupled discrete phase calculations.

coupled-calculations?
Select whether or not to couple continuous and discrete phase calculations.

implicit-momentum-coupling?
Enable/disable implicit treatment for the DPM momentum source terms.

implicit-source-term-coupling?
Enable/disable implicit treatment for all DPM source terms.

no.-of-cont-phase-iters-per-dpm-iter
Set the frequency with which the particle trajectory calculations are introduced.

reset-sources-at-timestep?
Enable/disable flush of DPM source terms at beginning of every time step.

underrelaxation-factor
Set the under-relaxation factor for the discrete phase sources.

update-dpm-sources-every-flow-iteration?
Enable/disable the update of DPM source terms every flow iteration (if this option is not enabled, the terms will be updated every DPM iteration).

numerics/
Enter the numerics menu to set numerical solution parameters.

`automated-scheme-selection?`
Enable/disable the adaptation of integration step length based on a maximum error.

`coupled-heat-mass-update`
Enable/disable coupled heat and mass update.

`drag-law`
Set the drag law.

`error-control?`
Adapt integration step length based on a maximum error.

`tracking-parameters`
Set parameters for the (initial) tracking step length.

`tracking-scheme`
Specify a tracking scheme.

`vaporization-limiting-factors`
Set the Vaporization Fractional Change Limits.

`options/`
Enter the options menu to set optional models.

`brownian-motion`
Enable/disable Brownian motion of particles.

`ensemble-average`
Ensemble average cloud properties.

`erosion-accretion`
Enable/disable erosion/accretion.

`film-separation-angle`
Set the angle between faces which causes film particles to separate from the wall.

`init-erosion-accretion-rate`
Initialize the erosion/accretion rates with zero.

`particle-radiation`
Enable/disable particle radiation.

`particle-staggering`
Enable/disable spatial and temporal staggering of the particle injections.

`saffman-lift-force`
Enable/disable Saffman lift force.

`step-report-sig-figures`
Set significant figures in the step-by-step report.

`thermophoretic-force`
Enable/disable thermophoretic force.

`track-in-absolute-frame`
Enable/disable tracking in absolute frame.

`two-way-coupling`
Enable/disable calculation of DPM sources in TKE equation.

`vaporization-options`
Set Vaporization options.

parallel/
Enter the parallel menu to set parameters for parallel DPM calculations.

enable-workpile?
Turn on/off particle workpile algorithm. This option is only available when the `define/models/dpm/parallel/use-shared-memory` option is selected.

n-threads
Set the number of processors to use for DPM. This option is only available when the `define/models/dpm/parallel/enable-workpile?` option is enabled.

report
Print particle workpile statistics. This option is only available when the `define/models/dpm/parallel/enable-workpile?` option is enabled.

use-hybrid
Specify that the calculations are performed using multicore cluster computing or shared-memory machines. This option works in conjunction with `openmpi` for a dynamic load balancing without migration of cells.

use-message-passing
Specify that the calculations are performed using cluster computing or shared-memory machines. With this option, the compute node processes themselves perform the particle work on their local partitions and particle migration to other compute nodes is implemented using message passing primitives.

use-shared-memory
Specify that the calculations are performed on shared-memory multiprocessor machines.

spray-model/
Enter the spray model menu.

breakup-model-summary
Current spray model settings.

droplet-breakup?
Enable/disable spray breakup model.

droplet-collision?
Enable/disable droplet collision model.

khrt-model
Enable KHRT breakup model.

number-of-breakup-parcels
Set the number of parcels to break up a droplet in the TAB model.

randomize-breakup-parcel-diameter?
Enable sampling of diameter for each TAB breakup parcel from a Rosin-Rammler distribution using a random number.

ssd-model
Enable SSD breakup model.

tab-model
Enable TAB breakup model.

wave-mass-cutoff
Set the minimum percentage of parent parcel mass shed before new parcel creation.

wave-model
Enable WAVE breakup model.

wave-spray-angle-constant
Set the spray-angle constant to compute orthogonal velocity components of child droplets after breakup.

unsteady-tracking
Enable/disable unsteady particle tracking.

user-defined
Set DPM user-defined functions.

energy?
Enable/disable the energy model.

eulerian-wallfilm/
Enter the Eulerian wall film model menu.

enable-wallfilm-model?
Enable/disable Eulerian Wall Film Model.

initialize-wallfilm-model
Initialize Eulerian Wall Film Model.

solve-wallfilm-equation?
Activate Eulerian Wall Film Equations.

model-options
Set Eulerian Wall Film Model Options.

film-material
Set Film Material and Properties.

solution-options
Set Eulerian Wall Film Model Solution Options.

frozen-flux?
Enable/disable frozen flux formulation for transient flows.

heat-exchanger/
Enter the heat exchanger menu.

dual-cell-model/
Enter the dual cell model menu.

add-heat-exchanger
Add heat-exchanger.

alternative-formulation?
Enable/disable alternative formulation for heat transfer calculations.

delete-heat-exchanger
Delete heat-exchanger.

heat-exchanger?
Enable/disable the dual cell heat-exchanger model.

modify-heat-exchanger
Modify heat-exchanger.

plot-NTU
Plot NTU vs. primary mass flow rate for each auxiliary mass flow rate.

write-NTU
Write NTU vs. primary mass flow rate for each auxiliary mass flow rate.

macro-model/
Enter the heat macro-model menu.

delete-heat-exchanger-group
Delete heat-exchanger group.

heat-exchanger?
Enable/disable heat-exchanger model.

heat-exchanger-group
Define heat-exchanger group.

heat-exchanger-macro-report
Report the computed values of heat rejection, outlet temperature, and inlet temperature for the macroscopic cells (macros) in a heat exchanger.

heat-exchanger-model
Define heat-exchanger core model.

heat-exchanger-report
Report the computed values of total heat rejection, outlet temperature, and inlet temperature for a specified heat-exchanger core.

heat-exchanger-zone
Specify the zone that represents the heat exchanger, the dimensions of the heat exchanger, the macro grid, and the coolant direction and properties.

plot-NTU
Plot NTU vs. primary mass flow rate for each auxiliary mass flow rate.

write-NTU
Write NTU vs. primary mass flow rate for each auxiliary mass flow rate.

multiphase/
Enter the multiphase model menu.

body-force-formulation
Specify body force formulation.

coupled-level-set
Enable coupled level set interface tracking method.

boiling-model-options
Specify the boiling model options. You can choose the `RPI boiling model`, `Non-equilibrium boiling`, or `Critical heat flux`.

eulerian-parameters
Specify Eulerian parameters.

mixture-parameters
Specify mixture parameters.

model
Specify multiphase model.

number-of-phases
Specify the number of phases.

options
Volume fraction parameters.

volume-fraction-parameters
Specify volume fraction parameters.

wet-steam/

Enter the wet steam model menu.

compile-user-defined-wetsteam-functions

Compile user-defined wet steam library.

enable?

Enable/disable the wet steam model.

load-unload-user-defined-wetsteam-library

Load or unload user-defined wet steam library.

set/

Enter the set menu for setting wet steam model options.

max-liquid-mass-fraction

Set the maximum limit on the condensed liquid-phase mass-fraction to prevent divergence.

noniterative-time-advance?

Enable/disable noniterative time advancement scheme.

nox?

Enable/disable the NOx model.

nox-parameters/

Enter the NOx parameters menu.

inlet-diffusion?

Enable/disable inclusion of diffusion at inlets.

nox-chemistry

Select NOx chemistry model.

nox-expert

Select additional NOx equations.

nox-turbulence-interaction

Set NOx turbulence interaction model.

radiation/

Enter the radiation models menu.

blending-factor

Set numeric option for Discrete Ordinate model. Make sure that **Second Order Upwind** is selected for the **Discrete Ordinates** spatial discretization for the `blending-factor` option to appear in the text command list.

discrete-ordinates?

Enable/disable discrete ordinates radiation model.

discrete-transfer?

Enable/disable discrete transfer radiation model.

do-coupling?

Enable/disable DO/energy coupling.

do-irradiation?

Enable/disable the DO irradiation model.

dtrm-parameters/

Enter the dtrm parameters menu.

check-ray-file

Read DTRM rays file.

controls
Set dtrm solution controls.

make-globs
Make globs (coarser mesh) for radiation.

ray-trace
Create DTRM rays for radiation.

fast-second-order-discrete-ordinate?
Enable/disable the fast-second-order option for Discrete Ordinate Model.

method-partially-specular-wall
Set the method for partially specular wall with discrete ordinate model.

non-gray-model-parameters
Set parameters for non-gray model.

p1?
Enable/disable P1 radiation model.

radiation-iteration-parameters
Set iteration parameters for radiation models.

radiation-model-parameters
Set parameters for radiation models.

rosseland?
Enable/disable Rosseland radiation model.

s2s?
Enable/disable S2S radiation model.

s2s-parameters/
Enter the S2S parameters menu.

compute-fpsc-values
Compute only fpsc values based on current settings

compute-vf-only
Compute/write view factors only.

compute-write-vf
Compute/write surface clusters and view factors for S2S radiation model.

non-participating-boundary-zones-temperature
Set temperature for the non-participating boundary zones.

print-thread-clusters
Print the following for all boundary threads: thread-id, number of faces, faces per surface cluster, and the number of surface clusters.

print-zonewise-radiation
Print the zonewise incoming radiation, viewfactors, and average temperature.

read-vf-file
Read S2S file.

set-global-faces-per-surface-cluster
Set global value of faces per surface cluster for all boundary zones.

set-vf-parameters
Set the parameters needed for the viewfactor calculations.

split-angle
Set split angle for the clustering algorithm.

use-new-cluster-algorithm
Use the new surface clustering algorithm.

use-old-cluster-algorithm
Use the old surface clustering algorithm.

solar?
Enable/disable solar model.

solar-calculator
Calculate sun direction and intensity.

solar-parameters/
Enter the solar parameters menu.

autoread-solar-data
Set autoread solar data parameters.

autosave-solar-data
Set autosave solar data parameters.

ground-reflectivity
Set ground reflectivity parameters.

illumination-parameters
Set illumination parameters.

iteration-parameters
Set update parameters.

quad-tree-parameters
Set quad-tree refinement parameters.

scattering-fraction
Set scattering fraction parameters.

sol-adjacent-fluidcells
Set solar load on for adjacent fluid cells.

sol-camera-pos
Set camera position based on sun direction vector.

sol-on-demand
Set solar load on demand.

sun-direction-vector
Set sun direction vector.

use-direction-from-sol-calc
Set direction computed from solar calculator.

solution-method-for-do-coupling
Enable/disable the solution method for DO/energy coupling.

wsggm-cell-based
Enable/disable WSGGM cell based method. Note that when enabled, the **wsggm-cell-based** option will become available in the **Absorption Coefficient** drop-down list in the **Create/Edit Materials** dialog box.

solidification-melting?
Enable/disable the solidification and melting model.

solver/
Enter the menu to select the solver.

density-based-explicit
Enable/disable the density-based-explicit solver.

density-based-implicit
Enable/disable the density-based-implicit solver.

pressure-based
Enable/disable the pressure-based solver.

soot?
Enable/disable the soot model.

soot-parameters/
Enter the soot parameters menu.

inlet-diffusion?
Enable/disable inclusion of diffusion at inlets.

modify-schmidt-number?
Change the turbulent Schmidt number for soot/nuclei equations.

soot-model-parameters
Select soot model parameters.

soot-process-parameters
Select soot process parameters.

soot-radiation-interaction
Enable/disable the soot-radiation interaction model.

soot-turbulence-interaction
Set soot-turbulence interaction model.

sox?
Enable/disable the SOx model.

sox-parameters/
Enter the SOx parameters menu.

inlet-diffusion?
Enable/disable inclusion of diffusion at inlets.

s-atom-balance?
Enable/disable S-atom mass balance calculation.

sox-chemistry
Select the SOx chemistry model.

sox-turbulence-interaction
Set the SOx /turbulence interaction model.

species/
Enter the species models menu.

CHEMKIN-CFD-from-Reaction-Design?
Enable/disable CHEMKIN-CFD from Reaction Design.

CHEMKIN-CFD-parameters/
Enter the expert CHEMKIN-CFD parameters menu.

add-cell-monitor
Monitor cell for debug output.

advanced-options
Set advanced parameter options.

basic-options
Set basic parameter options.

delete-cell-monitors
Delete cell monitors.

list-cell-monitors
List cell monitors.

clear-isat-table
Kill ISAT table.

coal-calculator
Set up coal modeling inputs.

decoupled-detailed-chemistry?
Enable/disable the Decoupled Detailed Chemistry model.

diffusion-energy-source?
Enable/disable diffusion energy source.

epdf-energy?
Enable/disable EPDF energy option.

flamelet-expert
Set flamelet expert parameters.

full-tabulation?
Enable/disable building of a full 2-mixture fraction table

heat-of-surface-reactions?
Enable/disable heat of surface reactions.

ignition-model?
Enable/disable the ignition model.

ignition-model-controls
Set ignition model parameters.

import-flamelet-for-restart
Import Flamelet File for Restart.

inert-transport-controls
Set inert transport model parameters.

inert-transport-model?
Enable/disable the inert transport model.

inlet-diffusion?
Enable/disable inclusion of diffusion at inlets.

integration-parameters
Set ISAT parameters.

init-unsteady-flamelet-prob
Initialize Unsteady Flamelet Probability.

liquid-energy-diffusion?
Enable/disable energy diffusion for liquid regime.

liquid-micro-mixing?
Enable/disable liquid micro mixing.

mass-deposition-source?
Enable/disable mass deposition source due to surface reactions.

mixing-model
Set PDF Transport mixing model.

multicomponent-diffusion?
Enable/disable multicomponent diffusion.

non-premixed-combustion?
Enable/disable non-premixed combustion model.

non-premixed-combustion-expert
Set PDF expert parameters.

non-premixed-combustion-parameters
Set PDF parameters.

off?
Enable/disable solution of species models.

partially-premixed-combustion?
Enable/disable partially premixed combustion model.

partially-premixed-combustion-expert
Set PDF expert parameters.

partially-premixed-combustion-parameters
Set PDF parameters.

particle-surface-reactions?
Enable/disable particle surface reactions.

pdf-transport?
Enable/disable the composition PDF transport combustion model.

pdf-transport-expert?
Enable/disable PDF Transport expert user.

premixed-model
Set premixed combustion model.

premixed-combustion?
Enable/disable premixed combustion model.

reaction-diffusion-balance?
Enable/disable reaction diffusion balance at reacting surface for surface reactions.

reacting-channel-model?
Enable/disable the Reacting Channel Model.

reacting-channel-model-options
Set Reacting Channel Model parameters.

relax-to-equil?
Enable/disable the Relaxation to Chemical Equilibrium model.

save-gradients?
Enable/disable storage of species mass fraction gradients.

set-premixed-combustion
Set premixed combustion parameters.

set-turb-chem-interaction
Set EDC model constants.

spark-model
Enable/disable spark model.

spark-model-controls
Set spark model parameters.

species-transport?
Enable/disable the species transport model.

stiff-chemistry?
Enable/disable stiff chemistry option.

surf-reaction-aggressiveness-factor?
Set the surface reaction aggressiveness factor.

surf-reaction-netm-params
Set the surface reaction parameters for the Non-Equilibrium Thermal Model.

thermal-diffusion?
Enable/disable thermal diffusion.

thickened-flame-model?
Enable/disable the Relaxation to Chemical Equilibrium model

volumetric-reactions?
Enable/disable volumetric reactions.

wall-surface-reactions?
Enable/disable wall surface reactions.

steady?
Enable/disable the steady solution model.

swirl?
Enable/disable axisymmetric swirl velocity.

unsteady-1st-order?
Enable/disable first-order unsteady solution model.

unsteady-2nd-order-bounded?
Enable/disable bounded second-order unsteady formulation.

unsteady-2nd-order?
Enable/disable the second-order unsteady solution model.

unsteady-global-time?
Enable/disable the unsteady global-time-step solution model.

viscous/
Enter the viscous model menu.

buoyancy-effects?
Enable/disable effects of buoyancy on turbulence.

curvature-correction?
Enable/disable the curvature correction.

des-limiter-option
Select the DES limiter option (F1, F2, none).

detached-eddy-simulation?
Enable/disable detached eddy simulation.

inviscid?
Enable/disable inviscid flow model.

ke-easm?
Enable/disable the EASM k- ε turbulence model.

ke-realizable?
Enable/disable the realizable k- ε turbulence model.

ke-rng?
Enable/disable the RNG k- ε turbulence model.

ke-standard?
Enable/disable the standard k- ε turbulence model.

k-kl-w?
Enable/disable the k-kl-ωturbulence model.

kw-compressibility?
Enable/disable the k- ω compressibility correction option.

kw-easm?
Enable/disable the EASM k- ω turbulence model.

kw-low-re-correction?
Enable/disable the k- ω low Re option.

kw-shear-correction?
Enable/disable the k- ω shear-flow correction option.

kw-sst?
Enable/disable the SST k- ω turbulence model.

kw-standard?
Enable/disable the standard k- ω turbulence model.

laminar?
Enable/disable laminar flow model.

large-eddy-simulation?
Enable/disable large eddy simulation.

les-subgrid-rng?
Enable/disable RNG subgrid-scale model.

les-subgrid-smagorinsky?
Enable/disable the Smagorinsky-Lilly subgrid-scale model.

les-subgrid-tke?
Enable/disable kinetic energy transport subgrid-scale model.

les-subgrid-wale?
Enable/disable WALE subgrid-scale model.

`les-subgrid-wmles?`
Enable/disable the WMLES subgrid-scale model.

`mixing-length?`
Enable/disable mixing-length (algebraic) turbulence model.

`multiphase-turbulence/`
Enter the multiphase turbulence menu.

`multiphase-options`
Enable/disable multiphase options.

`rsm-multiphase-models`
Select Reynolds Stress multiphase model.

`turbulence-multiphase-models`
Select k- ε multiphase model.

`near-wall-treatment/`
Enter the near wall treatment menu.

`enhanced-wall-treatment?`
Enable/disable enhanced wall functions.

`non-equilibrium-wall-fn?`
Enable/disable non-equilibrium wall functions.

`scalable-wall-functions?`
Enable/disable scalable wall functions.

`standard-wall-fn?`
Enable/disable standard wall functions.

`user-defined-wall-functions?`
Enable/disable user-defined wall functions.

`werner-wengle-wall-fn?`
Enable/disable Werner-Wengle wall functions.

`wf-pressure-gradient-effects?`
Enable/disable wall function pressure- gradient effects.

`wf-thermal-effects?`
Enable/disable wall function thermal effects.

`reynolds-stress-model?`
Enable/disable the Reynolds-stress turbulence model.

`rng-differential-visc?`
Enable/disable the differential-viscosity model.

`rng-swirl-model?`
Enable/disable swirl corrections for rng-model.

`rsm-linear-pressure-strain?`
Enable/disable the linear pressure-strain model in RSM.

`rsm-omega-based?`
Enable/disable the low-Reynolds-Stress-omega model.

`rsm-solve-tke?`
Enable/disable the solution of T.K.E. in RSM model.

`rsm-ssg-pressure-strain?`
Enable/disable quadratic pressure-strain model in RSM.

`rsm-wall-echo?`
Enable/disable wall-echo effects in RSM model.

`sa-alternate-prod?`
Enable/disable strain/vorticity production in Spalart-Allmaras model.

`sa-damping?`
Enable/disable full low-Reynolds number form of Spalart-Allmaras model.

Note

This option is only available if your response was `no` to `sa-enhanced-wall-treatment?`.

`sa-enhanced-wall-treatment?`
Enable/disable the enhanced wall treatment for the Spalart-Allmaras model. If disabled, no smooth blending between the viscous sublayer and the log-law formulation is employed, as was done in versions previous to FLUENT14.

`sas?`
Enable/disable the SAS turbulence model.

`spalart-allmaras?`
Enable/disable Spalart-Allmaras turbulence model.

`transition-sst?`
Enable/disable the transition SST turbulence model.

`trans-sst-roughness-correlation?`
Enable/disable the Transition-SST roughness correlation option.

`turbulence-expert/`
Enter the turbulence expert menu.

`curvature-correction-coefficient`
Set the strength of the curvature correction term. The default value is 1. This is available after the `curvature-correction?` option is enabled.

`kato-launder-model?`
Enable/disable Kato-Launder modification.

`kw-vorticity-based-production?`
Enable/disable vorticity based production.

`low-re-ke?`
Enable/disable the low-Re k- ε turbulence model.

`low-re-ke-index`
Select which low-Reynolds-number k- ε model is to be used. Six models are available:

Index	Model
0	Abid
1	Lam-Bremhorst
2	Launder-Sharma

Index	Model
3	Yang-Shih
4	Abe-Kondoh-Nagano
5	Chang-Hsieh-Chen

Contact your ANSYS, Inc. technical support engineer for more details.

`non-newtonian-modification?`
Enable/disable non-Newtonian modification for Lam-Bremhorst model.

`restore-sst-v61?`
Enable/disable SST formulation of v6.1.

`rke-cmu-rotation-term?`
Modify the C_μ definition for the realizable k- ε model.

Important

Note that the use of the realizable k- ε model with multiple reference frames is not recommended. This text command is provided for expert users who want to experiment with this combination of models. Others should use it only on the advice of a technical support engineer.

`thermal-p-function?`
Enable/disable Jayatilleke P function.

`turb-non-newtonian?`
Enable/disable turbulence for non-Newtonian fluids.

`turbulence-damping?`
Enable/disable turbulence damping and set turbulence damping parameters.

`turb-pk-compressible?`
Enable/disable turbulent production due to compressible divergence.

`user-defined`
Set user-defined functions related to turbulent viscosity.

`user-defined-transition`
Set user-defined transition correlations.

`v2f?`
Enable/disable V2F turbulence model.

`zero-equation-hvac?`
Enable/disable zero-equation HVAC turbulence model.

`operating-conditions/`
Enter the define operating conditions menu.

`gravity`
Set gravitational acceleration.

`operating-density?`
Enable/disable use of a specified operating density.

operating-pressure
Set the operating pressure.

operating-temperature
Set the operating temperature for Boussinesq.

reference-pressure-location
Set the location of the cell whose pressure value is used to adjust the gauge pressure field for incompressible flows that do not involve any pressure boundaries.

used-ref-pressure-location
See the actual coordinates of the reference pressure used.

use-inlet-temperature-for-operating-density
Use inlet temperature to calculate operating density.

parameters/
Enter the parameters menu.

enable-in-TUI?
Enable/disable parameters in the boundary conditions text user interface.

input-parameters/
Enter the input-parameters menu.

delete
Delete an input parameter.

edit
Edit an input parameter.

output-parameters/
Enter the output-parameters menu.

create
Create an output parameter.

delete
Delete an output parameter.

edit
Edit an output parameter.

print-all-to-console
Display all parameter values in the console.

print-to-console
Display parameter value in the console.

write-all-to-file
Write all parameter values to file.

write-to-file
Write parameter value to file.

periodic-conditions/
Enter the periodic conditions menu.

massflow-rate-specification?
Enable/disable specification of mass flow rate at the periodic boundary.

pressure-gradient-specification?
Enable/disable specification of pressure gradient at the periodic boundary.

phases/
Enter the phases menu.

domain/
Enter the domain menu.

iac-expert/
Enter the IAC expert setting menu.

hibiki-ishii-model
Set HI model coefficients

ishii-kim-model
Set IK model coefficients

yao-morel-model
set ym model coefficients

interaction-domain
Set models and properties for a domain of this type.

phase-domain
Set models and properties for a domain of this type.

profiles/
Enter the boundary profiles menu.

delete
Delete a profile.

delete-all
Delete all boundary-profiles.

interpolation-method
Choose the method for interpolation of profiles.

list-profiles
List all profiles.

list-profile-fields
List the fields of a particular profile.

morphing?
Enable/disable profile morphing options in Orient Profile panel.

update-interval
Set interval between updates of dynamic profiles.

solution-strategy/
Enter the automatic initialization and case modification strategy menu.

automatic-case-modification/
Enter the automatic case modification menu.

before-init-modification
Specify modification to be performed before initialization.

modifications
Specify modifications to be performed during solution.

original-settings
Specify modification to be performed after initialization to restore to original settings.

`automatic-initialization`
Define how the case is to be automatically initialized.

`continue-strategy-execution`
Continue execution of the currently defined automatic initialization and case modification strategy.

`enable-strategy?`
Enable/disable automatic initialization and case modification.

`execute-strategy`
Execute the currently defined automatic initialization and case modification strategy.

`turbo/`
Enter the turbo menu.

`define-topology`
Define a turbo topology.

`mesh-method`
Set turbo structured mesh generation method.

`search-method`
Set search method for a topology.

`projection-method`
Set 2D projection method.

`units`
Set unit conversion factors.

`user-defined/`
Enter the user-defined functions and scalars menu.

`1D-coupling`
Load 1D library.

`compiled-functions`
Open user-defined function library.

`execute-on-demand`
Execute UDFs on demand.

`fan-model`
Configure user-defined fan model.

`function-hooks`
Hook up user-defined functions.

`interpreted-functions`
Load interpreted user-defined functions.

`real-gas-models`
Enter the real-gas menu to enable/configure real gas model.

`nist-multispecies-real-gas-model`
Load the NIST real-gas library.

`nist-real-gas-model`
Load the NIST real-gas library.

`set-phase`
Select the phase for NIST real gas model.

user-defined-multispecies-real-gas-model
Load a user-defined multispecies real-gas library.

user-defined-real-gas-model
Load the user-defined real-gas library.

use-contributed-cpp?
Enable/disable use of cpp from the Fluent.Inc/contrib directory.

user-defined-memory
Allocate user-defined memory.

user-defined-scalars
Define user-defined scalars.

Chapter 3: display/

`add-custom-vector`
Add new custom vector definition.

`annotate`
Add annotation text to a graphics window. It will prompt you for a string to use as the annotation text, and then a dialog box will prompt you to select a screen location using the mouse-probe button on your mouse.

`clear-annotations`
Remove all annotations and attachment lines from the active graphics window.

`close-window`
Close a graphics window.

`contour`
Prompts for a scalar field and minimum and maximum values, and then displays a contour plot.

`display-custom-vector`
Display custom vector.

`flamelet-data`
Display flamelet data.

`carpet-plot`
Enable/disable display of carpet plot of a property.

`draw-number-box?`
Enable/disable display of the numbers box.

`plot-1d-slice?`
Enable/disable plot of the 1D-slice.

`write-to-file?`
Enable/disable writing the 1D-slice to file instead of plot.

`graphics-window-layout`
Arrange the graphics window layout.

`mesh`
Display the entire mesh. For 3D, you will be asked to confirm that you really want to draw the entire mesh (not just the `mesh-outline`).

`mesh-outline`
Display the mesh boundaries.

`mesh-partition-boundary`
Display mesh partition boundaries.

`multigrid-coarsening`
Display a coarse mesh level from the last multigrid coarsening.

`open-window`
Open a graphics window.

`particle-tracks/`
Enter the particle tracks menu.

`particle-tracks`
Calculate and display particle tracks from defined injections.

`plot-write-xy-plot`
Plot or write an XY plot of particle tracks.

`path-lines/`
Enter the pathlines menu.

`path-lines`
Display pathlines from a surface.

`plot-write-xy-plot`
Plot or write an XY plot of pathlines.

`write-to-files`
Write pathlines to a file.

`pdf-data/`
Enter the PDF data menu.

`carpet-plot`
Enable/disable the display of a carpet plot of a property.

`draw-number-box?`
Enable/disable the display of the numbers box.

`plot-1d-slice?`
Enable/disable a plot of the 1D-slice.

`write-to-file?`
Enable/disable writing the 1D-slice to file instead of plot.

`plot-reacting-channel-vars`
Plot the reacting channel variables.

`profile`
Display profiles of a flow variable.

`re-render`
Re-render the last contour, profile, or vector plot with updated surfaces, meshed, lights, colormap, rendering options, etc., without recalculating the contour data.

`re-scale`
Re-render the last contour, profile, or vector plot with updated scale, surfaces, meshes, lights, colormap, rendering options, etc., but without recalculating the field data.

`save-picture`
Generate a "hardcopy" of the active window.

`set/`
Enter the set menu to set display parameters.

`color-map/`
Enter the color map menu, which contains names of predefined and user-defined (in the **Colormap Editor** panel) colormaps that can be selected. It prompts you for the name of the colormap to be used.

`colors/`
Enter the color options menu.

background
Set the background (window) color.

color-by-type?
Determine whether to color meshes by type or by ID.

color-scheme
Set the color scheme. You can choose to display your graphics in the classic ANSYS FLUENT color scheme, or you can use the default Workbench color scheme.

axis-faces
Set the color of axisymmetric faces.

far-field-faces
Set the color of far field faces.

free-surface-faces
Set the color of free-surface faces.

foreground
Set the foreground (text and window frame) color.

highlight-color
Set highlight color.

inlet-faces
Set the color of inlet faces.

interface-faces
Set the color of mesh interfaces.

interior-faces
Set the color of interior faces.

internal-faces
Set the color of internal interface faces.

outlet-faces
Set the color of outlet faces.

periodic-faces
Set the color of periodic faces.

rans-les-interface-faces
Set the color of RANS/LES interface faces.

symmetry-faces
Set the color of symmetric faces.

traction-faces
Set the color of traction faces.

wall-faces
Set the color of wall faces.

list
List available colors.

reset-colors
Reset individual mesh surface colors to the defaults.

skip-label
Set the number of labels to be skipped in the colormap scale.

surface
Set the color of surfaces.

contours/
Enter the contour options menu.

clip-to-range?
Turn the clip to range option for filled contours on/off.

filled-contours?
Turn the filled contours option on/off (deselects `line-contours?`).

global-range?
Turn the global range for contours on/off.

line-contours?
Turn the line contours option on/off (deselects `filled-contours?`).

log-scale?
Specify a decimal or logarithmic color scale for contours.

n-contour
Set the number of contour levels.

node-values?
Set the option to use scalar field at nodes when computing the contours.

render-mesh?
Determine whether or not to render the mesh on top of contours, vectors, etc.

surfaces
Set the surfaces on which contours are drawn. You can include a wildcard (*) within the surface names.

duplicate-node-display?
Enable/disable the display of duplicate nodes in a mesh.

element-shrink
Set shrinkage of both faces and cells. A value of zero indicates no shrinkage, while a value of one will shrink each face or cell to a point.

filled-mesh?
Determine whether the meshes are drawn as wireframe or solid.

mesh-level
Set coarse mesh level to be drawn.

mesh-partitions?
Enable/disable option to draw mesh partition boundaries.

mesh-surfaces
Set surface IDs to be drawn as meshes. You can include a wildcard (*) within the surface names.

mesh-zones
Set zone IDs to be drawn as meshes.

picture/
Enter the `save-picture` options menu.

color-mode/
Enter the hardcopy/save-picture color mode menu.

color
Plot hardcopies in color.

gray-scale
Convert color to grayscale for hardcopy.

list
Display the current hardcopy color mode.

mono-chrome
Convert color to monochrome (black and white) for hardcopy.

dpi
Set the resolution for EPS and Postscript files; specifies the resolution in dots per inch (DPI) instead of setting the width and height.

driver/
Enter the set hardcopy driver menu.

dump-window
Set the command used to dump the graphics window to a file.

eps
Produce encapsulated PostScript (EPS) output for hardcopies.

jpeg
Produce JPEG output for hardcopies. (This is the default file type.)

list
List the current hardcopy driver.

options
Set the hardcopy options. Available options are: "no gamma correction", disables gamma correction of colors; "pen speed = f", where f is a real number in [0,1]; "physical size = (width, height)", where width and height are the actual measurements of the printable area of the page in centimeters; "subscreen = (left, right, bottom, top)", where left, right, bottom, and top are numbers in [-1,1] describing a subwindow on the page in which to place the hardcopy. The options may be combined by separating them with commas. The pen speed option is only meaningful to the HPGL driver.

png
Use PNG output for hardcopies.

post-format/
Enter the PostScript driver format menu.

fast-raster
Enable a raster file that may be larger than the standard raster file, but will print much more quickly.

raster
Enable the standard raster file.

rle-raster
Enable a run-length encoded raster file that will be about the same size as the standard raster file, but will print slightly more quickly.

vector
Enable the standard vector file.

post-script
Produce PostScript output for hardcopies.

ppm
Produce PPM output for hardcopies.

`tiff`
Produce TIFF output for hardcopies.

`vrml`
Use VRML output for hardcopies.

`invert-background?`
Exchange foreground/background colors for hardcopy.

`landscape?`
Plot hardcopies in landscape or portrait orientation.

`preview`
Apply the settings of the color-mode, invert-background, and landscape options to the currently active graphics window to preview the appearance of printed hardcopies.

`x-resolution`
Set the width of raster-formatted images in pixels (0 implies current window size).

`y-resolution`
Set the height of raster-formatted images in pixels (0 implies current window size).

`lights/`
Enter the lights menu.

`headlight-on?`
Turn the light that moves with the camera on or off.

`lighting-interpolation`
Set lighting interpolation method.

`flat`
Use flat shading for meshes and polygons.

`gouraud`
Use Gouraud shading to calculate the color at each vertex of a polygon and interpolates it in the interior.

`phong`
Use Phong shading to interpolate the normals for each pixel of a polygon and computes a color at every pixel.

`lights-on?`
Turn all active lighting on/off.

`set-ambient-color`
Set the ambient color for the scene. The ambient color is the background light color in a scene.

`set-light`
Add or modify a directional, colored light.

`line-weight`
Set the line-weight factor for the window.

`marker-size`
Set the size of markers used to represent points.

`marker-symbol`
Set the type of markers used to represent points.

`mirror-zones`
Set the zones about which the domain is mirrored (symmetry planes).

mouse-buttons
Prompts you to select a function for each of the mouse buttons.

mouse-probes?
Enable/disable mouse probe capability.

n-stream-func
Set number of iterations used in computing stream function.

overlays?
Enable/disable overlays.

particle-tracks/
Enter the particle-tracks menu to set parameters for display of particle tracks.

arrow-scale
Set the scale factor for arrows drawn on particle tracks.

arrow-space
Set the spacing factor for arrows drawn on particle tracks.

coarsen-factor
Set the coarsening factor for particle tracks.

display?
Determine whether particle tracks shall be displayed or only tracked.

filter-settings/

Set filter for particle display.

enable-filtering?
Specify whether particle display is filtered.

filter-variable
Select a variable used for filtering of particles.

inside?
Specify whether filter variable needs to be inside min/max to be displayed (else outside min/max).

maximum
Specify the upper bound for the filter variable.

minimum
Specify the lower bound for the filter variable.

history-filename
Specify the name of the particle history file.

line-width
Set the width for particle track.

marker-size
Set the size of markers used to represent particle tracks.

particle-skip
Specify how many particle tracks should be displayed.

radius
Set the radius for particle track (ribbon/cylinder only) cross section.

report-to
Specify the destination for the report (console, file, none).

report-type
Set the report type for particle tracks.

report-variables
Set the report variables.

report-default-variables
Set the report variables to default.

sphere-attrib
Specify the size and number of slices to be used in drawing spheres.

sphere-settings/

Set filter for particle display.

auto-range?
Specify whether displayed spheres should include auto range of variable to size spheres.

diameter
Diameter of the spheres when `vary-diameter` is disabled.

maximum
Set the maximum value of the sphere to be displayed.

minimum
Set the minimum value of the sphere to be displayed.

scale-factor
Specify a scale factor to enlarge/reduce the size of spheres.

size-variable
Select a particle variable to size the spheres.

smooth-parameter
Specify number of slices to be used in drawing spheres.

vary-diameter?
Specify whether the spheres can vary with another variable.

style
Set the display style for particle track (line/ribbon/cylinder/sphere).

track-single-particle-stream?
Specify the stream ID to be tracked.

twist-factor
Set the scale factor for twisting (ribbons only).

vector-settings/

Set vector specific input.

color-variable?
Specify whether the vectors should be colored by variable specified in /display/particle-track/particle-track (if false use a constant color).

constant-color
Specify a constant color for the vectors.

length-to-head-ratio
Specify ratio of length to head for vectors and length to diameter for cylinders.

length-variable?
Specify whether the displayed vectors have length varying with another variable.

scale-factor
Specify a scale factor to enlarge/reduce the length of vectors.

style
Enable and set the display style for particle vectors (none/vector/centered-vector/centered-cylinder).

vector-length
Specify the length of constant vectors.

vector-length-variable
Select a particle variable to specify the length of vectors.

vector-variable
Select a particle vector function to specify vector direction.

path-lines/
Set parameters for display of pathlines.

arrow-scale
Set the scale factor for arrows drawn on pathlines.

arrow-space
Set the spacing factor for arrows drawn on pathlines.

display-steps
Set the display stepping for pathlines.

error-control?
Set error control during pathline computation.

line-width
Set the width for pathlines.

marker-size
Set the marker size for particle drawing.

maximum-error
Set the maximum error allowed while computing the pathlines.

maximum-steps
Set the maximum number of steps to take for pathlines.

radius
Set the radius for pathline (ribbons/cylinder only) cross-section.

relative-pathlines?
Enable/disable the tracking of pathlines in a relative coordinate system.

reverse?
Set direction of path tracking.

sphere-attrib
Specify the size and number of slices to be used in drawing spheres.

step-size
Set the step length between particle positions for pathlines.

style
Select the pathline style (line, point, ribbon, triangle, cylinder).

time-step
Set the time step between particle positions for pathlines.

twist-factor
Set the scale factor for twisting (ribbons only).

periodic-repeats
Set number of periodic repetitions.

proximity-zones
Set zones to be used for boundary cell distance and boundary proximity.

render-mesh?
Enable/disable rendering the mesh on top of contours, vectors, etc.

rendering-options/
Enter the rendering options menu, which contains the commands that allow you to set options that determine how the scene is rendered.

animation-option
Use of `wireframe` or `all` during animation.

auto-spin?
Enable/disable mouse view rotations to continue to spin the display after the button is released.

color-map-alignment
Set the color bar alignment.

device-info
Print out information about your graphics driver.

double-buffering?
Enable/disable double buffering. Double buffering dramatically reduces screen flicker during graphics updates. If your display hardware does not support double buffering and you turn this option on, double buffering will be done in software. Software double buffering uses extra memory.

driver/
Change the current graphics driver.

gl
IRIS GL

null
No graphics driver.

opengl
OpenGL

pex
X11 PEX

hbx
HP Starbase

x11
X11

xgl
Sun XGL

msw
Microsoft Windows

face-displacement
Set face displacement value in Z-buffer units along the Camera Z-axis.

hidden-lines?
Turn hidden line removal on/off.

hidden-surfaces?
Turn hidden surface removal on/off.

hidden-surface-method/
Allows you to choose from among the hidden surface removal methods that ANSYS FLUENT supports. These options (listed below) are display hardware dependent.

hardware-z-buffer
is the fastest method if your hardware supports it. The accuracy and speed of this method is hardware dependent.

painters
will show less edge-aliasing effects than hardware-z- buffer. This method is often used instead of software-z-buffer when memory is limited.

software-z-buffer
is the fastest of the accurate software methods available (especially for complex scenes), but it is memory intensive.

z-sort-only
is a fast software method, but it is not as accurate as software-z-buffer.

outer-face-cull?
Enable/disable discarding outer faces during display.

set-rendering-options
Set the rendering options.

surface-edge-visibility
Set edge visibility flags for surfaces.

reset-graphics
Reset the graphics system.

title
Set problem title.

left-top
Set the title text for left top in title segment.

left-bottom
Set the title text for left bottom in title segment.

right-top
Set the title text for right top in title segment.

right-middle
Set the title text for right middle in title segment.

right-bottom
Set the title text for right bottom in title segment.

velocity-vectors/
Enter the menu to set parameters for display of velocity vectors.

auto-scale?
Auto-scale all vectors so that vector overlap is minimal.

`color`
Set the color of all velocity vectors to the color specified. The color scale is ignored. This is useful when overlaying a vector plot over a contour plot.

`color-levels`
Set the number of colors used from the colormap.

`component-x?`
Set the option to use only the x component of the velocity vectors during display.

`component-y?`
Set the option to use only the y component of the velocity vectors during display.

`component-z?`
Set the option to use only the z component of the velocity vectors during display.

`constant-length?`
Set the option to draw velocity vectors of constant length. This shows only the direction of the velocity vectors.

`global-range?`
Turn global range for vectors on/off.

`in-plane?`
Toggle the display of velocity vector components in the plane of the surface selected for display.

`log-scale?`
Toggle whether color scale is logarithmic or linear.

`node-values?`
Enable/disable the plotting of node values. Cell values will be plotted if "no".

`relative?`
Toggle the display of relative velocity vectors.

`render-mesh?`
Enable/disable rendering the mseh on top of contours, vectors, etc.

`scale`
Set the value by which the vector length will be scaled.

`scale-head`
Set the value by which the vector head will be scaled.

`surfaces`
Set surfaces on which vectors are drawn. You can include a wildcard (*) within the surface names.

`windows/`
Enter the windows option menu, which contains commands that allow you to customize the relative positions of subwindows inside the active graphics window.

`aspect-ratio`
Set the aspect ratio of the active window.

`axes/`
Enter the axes window options menu.

`border?`
Set whether to draw a border around the axes window.

`bottom`
Set the bottom boundary of the axes window.

clear?
Set the transparency of the axes window.

left
Set the left boundary of the axes window.

right
Set the right boundary of the axes window.

top
Set the top boundary of the axes window.

visible?
Turn axes visibility on/off.

main/
Enter the main view window options menu.

border?
Set whether or not to draw a border around the main viewing window.

bottom
Set the bottom boundary of the main viewing window.

left
Set the left boundary of the main viewing window.

right
Set the right boundary of the main viewing window.

top
Set the top boundary of the main viewing window.

visible?
Turn visibility of the main viewing window on/off.

scale/
Enter the color scale window options menu.

alignment
Set the colormap position to the bottom, left, top, or right.

border?
Set whether or not to draw a border around the color scale window.

bottom
Set the bottom boundary of the color scale window.

clear?
Set the transparency of the scale window.

format
Set the number format of the color scale window. (e.g., %0.2e)

font-size
Set the font size of the color scale window.

left
Set the left boundary of the color scale window.

margin
Set the margin of the color scale window.

`right`
Set the right boundary of the color scale window.

`top`
Set the top boundary of the color scale window.

`visible?`
Turn visibility of the color scale window on/off.

`text/`
Enter the text window options menu.

`application?`
Show/hide the application name in the picture.

`border?`
Set whether or not to draw a border around the text window.

`bottom`
Set the bottom boundary of the text window.

`clear?`
Enable/disable text window transparency.

`company?`
Show/hide the company name in the picture.

`date?`
Show/hide the date in the picture.

`left`
Set the left boundary of the text window.

`right`
Set the right boundary of the text window.

`top`
Set the top boundary of the text window.

`visible?`
Turn visibility of the text window on/off.

`video/`
Enter the video window options menu.

`background`
Set the background color of the graphics window. The color is specified as a string of three comma-separated numbers between 0 and 1, representing red, green, and blue. For example, to change the background from black (default) to gray, you would enter `".5,.5,.5"` after selecting the `background` command.

`color-filter`
Set the video color filter. For example, to change the color filter from its default setting to PAL video with a saturation of 80% and a brightness of 90%, you would enter `"video=pal,sat=.8,gain=.9"` after selecting the `color-filter` command.

`foreground`
Set the foreground (text) color of the graphics window. The color is specified as a string of three comma-separated numbers between 0 and 1, representing red, green, and blue. For example, to change the foreground from white (default) to gray, you would enter `".5,.5,.5"` after selecting the `foreground` command.

on?
Enable/disable video picture settings.

pixel-size
Set the window size in pixels.

xy/
Enter the XY plot window options menu.

border?
Set whether or not to draw a border around the XY plot window.

bottom
Set the bottom boundary of the XY plot window.

left
Set the left boundary of the XY plot window.

logo?
Enable/disable visibility of the logo in graphics window.

right
Set the right boundary of the XY plot window.

top
Set the top boundary of the XY plot window.

visible?
Turn visibility of the XY plot window on/off.

duplicate-node-display?
Set a flag to remove the duplicate nodes in the mesh display.

zero-angle-dir
Set the vector having zero angular coordinates.

set-window
Set a graphics window to be the active window.

surface/
Enter the data surface-manipulation menu. For a description of the items in this menu, see *surface/* (p. 97).

surface-cells
Draw the cells on the specified surfaces. You can include a wildcard (*) within the surface names.

surface-mesh
Draw the mesh defined by the specified surfaces. You can include a wildcard (*) within the surface names.

update-scene/
Enter the scene options menu.

delete
Delete selected geometries.

display
Display selected geometries.

draw-frame?
Enable/disable drawing the bounding frame.

iso-sweep
Change iso-sweep values.

`overlays?`
Enable/disable the overlays option.

`pathline`
Change pathline attributes.

`select-geometry`
Select geometry to be updated.

`set-frame`
Change frame options.

`time`
Change time-step value.

`transform`
Apply transformation matrix on selected geometries.

`vector`
Display vectors of a space vector variable.

`velocity-vector`
Prompts for a scalar field by which to color the vectors, the minimum and maximum values, and the scale factor, and then draws the velocity vectors.

`view/`
Enter the view manipulation menu. For a description of the items in this menu, see *`views/`* (p. 101).

`zone-mesh`
Draw the mesh defined by specified face zones. Zone names can be indicated using wildcards (*).

Chapter 4: exit / close-fluent

`exit`

Exit program.

`close-fluent`

(ANSYS FLUENT in Workbench only) Exit program.

Chapter 5: file/

`async-optimize?`
Choose whether to optimize file I/O using scratch disks and asynchronous operations.

`auto-save/`
Enter the auto save menu.

`append-file-name-with`
Set the suffix for auto-saved files. The file name can be appended by flow-time, time-step value, or by user-specified flags in file name.

`case-frequency`
Specify the frequency (in iterations or time steps) with which case files are saved.

`data-frequency`
Specify the frequency (in iterations or time steps) with which data files are saved.

`max-files`
Set the maximum number of files. Once the maximum is reached, files will be erased as new files are written.

`overwrite-existing-files`
Overwrite existing files when files are automatically saved.

`retain-most-recent-files`
Set autosave to retain the 5 most recent files.

`root-name`
Specify the root name for the files that are saved.

`binary-files?`
Indicate whether to write binary or text format case and data files.

`confirm-overwrite?`
Confirm attempts to overwrite existing files.

`data-file-options`
Set derived quantities to be written in data file.

`define-macro`
Save input to a named macro.

`em-mapping`
Enter the electromagnetic loss mapping menu.

Important

The `em-mapping` option is only available in serial and parallel ANSYS FLUENT. It is available in ANSYS FLUENT under Workbench only when there is a connection detected between the ANSYS FLUENT and Ansoft Maxwell applications (and only the `volumetric-energy-loss` command is available).

`maintain-loss-on-initialization`
Maintain the loss data provided by Ansoft even if solution is initialized.

`remove-loss-only`
Remove the loss data provided by Ansoft and keep all other solution data.

`volumetric-energy-loss`
Maps the total loss (i.e., heat source) from Ansoft Maxwell to ANSYS FLUENT so that you can perform a thermal analysis. This option is only available when there is a connection detected between the ANSYS FLUENT and Ansoft Maxwell applications.

`execute-macro`
Run a previously defined macro.

`export-to-cfd-post`
Export data files that are compatible with CFD-Post (i.e., `.cdat` and `.cst` files) and open CFD-Post, if desired.

`export/`
Export case and data information.

`abaqus`
Write an ABAQUS file.

`ascii`
Write an ASCII file.

`avs`
Write an AVS UCD file.

`cfd-post-compatible`
Write data files that are compatible with CFD-Post (i.e., `.cdat` and `.cst` files).

`cgns`
Write a CGNS file.

`custom-heat-flux`
Write a generic file for heat transfer.

`dx`
Write an IBM Data Explorer format file.

`ensight`
Write EnSight geometry, velocity, and scalar files.

`ensight-gold`
Write EnSight Gold geometry, velocity, and scalar files.

`ensight-gold-transient`
Write EnSight Gold Transient geometry, velocity, and scalar files.

`fast-mesh`
Write FAST/Plot3D unstructured mesh file.

`fast-scalar`
Write FAST/Plot3D unstructured scalar function file.

`fast-solution`
Write FAST/Plot3D unstructured solution file.

`fast-velocity`
Write FAST/Plot3D unstructured vector function file.

fieldview
Write FIELDVIEW case and data files.

fieldview-data
Write FIELDVIEW case and data files.

fieldview-unstruct
Write FIELDVIEW unstructured combined file.

fieldview-unstruct-mesh
Write FIELDVIEW unstructured mesh-only file.

fieldview-unstruct-data
Write FIELDVIEW unstructured results-only file.

gambit
Write GAMBIT neutral file.

icemcfd-for-icepak
Write a binary ICEMCFD domain file.

ideas
Write an I-deas universal file.

mechanical-apdl
Write a Mechanical APDL file.

mechanical-apdl-input
Write a Mechanical APDL Input file.

nastran
Write a NASTRAN file.

particle-history-data
Export particle-history data.

patran-neutral
Write a PATRAN neutral file.

patran-nodal
Write a PATRAN nodal results file.

radtherm
Export RADTHERM file.

tecplot
Write a Tecplot+3DV format file.

fsi/
Enter the fluid-structure interaction menu.

display-fsi-mesh
Display the mesh for a fluid-structure interaction.

read-fsi-mesh
Read an FEM mesh for one-way data mapping from ANSYS FLUENT.

write-fsi-mesh
Write a fluid-structure interaction mesh file.

import/
Import case and data information.

abaqus/
Import an ABAQUS file.

fil
Read an ABAQUS .fil result file as a case file.

input
Read an ABAQUS input file as a case file.

odb
Read an ABAQUS odb file as a case file.

cfx/
Import a CFX file.

definition
Read a CFX definition file as a case file.

result
Read a CFX definition file as a case file.

cgns/
Import a CGNS file.

data
Read data from CGNS file.

mesh
Import a CGNS mesh file.

mesh-data
Import a CGNS mesh file and data file.

chemkin-mechanism
Read a CHEMKIN mechanism file.

chemkin-report-each-line?
Enable/disable reporting after reading each line.

ensight
Read an EnSight file as a case file.

fidap
Import a FIDAP neutral file.

flamelet/
Import a flamelet file.

standard
Read a standard format flamelet file.

oppdif
Read an OPPDIF format flamelet file.

cfx-rif
Read a CFX-RIF format flamelet file.

fluent4-case
Import a formatted ANSYS FLUENT 4 case file.

gambit
Import a GAMBIT neutral file.

hypermesh
Read a HYPERMESH file as a case file.

ic3m
Read IC3M files.

ideas-universal
Import an I-deas Universal file.

lstc/
Import an LSTC file.

input
Read an LSTC input file as a case file.

state
Read an LSTC result file as a case file.

marc-post
Read a MARC POST file as a case file.

mechanical-apdl/
Import a Mechanical APDL file.

input
Read a Mechanical APDL file as a case file.

result
Read a Mechanical APDL result file as a case file.

nastran/
Import a NASTRAN file.

bulkdata
Read a NASTRAN file as a case file.

output2
Read a NASTRAN op2 file as a case file.

partition/
Enter the partition menu to set conditions for partitioning an ANSYS FLUENT case file during read.

metis
Read and partition an ANSYS FLUENT case file.

metis-zone
Read and partition an ANSYS FLUENT case file.

patran/
Import a PATRAN neutral file (zones defined by named components).

neutral
Read a PATRAN Neutral file (zones defined by named components) as a case file.

plot3d/
Import a PLOT3D file.

mesh
Read a PLOT3D file as a case file.

tecplot
Enter the Tecplot menu.

mesh
Read a Tecplot binary file as a case file.

prebfc-structured
Import a formatted PreBFC structured mesh file.

ptc-mechanica
Read a PTC Mechanica Design file as a case file.

interpolate/
Interpolate data to/from another grid.

read-data
Read and interpolate data.

write-data
Write data for interpolation.

zone-selection
Define a list of cell zone IDs. If specified, interpolation data will be read/written for these cell zones only.

read-case
Read a case file.

read-case-data
Read a case and a data file.

read-data
Read a data file.

read-field-functions
Read custom field function definitions from a file.

read-injections
Read all DPM injections from a file.

read-isat-table
Read ISAT Table.

read-journal
Read command input from a file.

read-macros
Read macro definitions from a file.

read-pdf
Read a PDF file.

read-profile
Read boundary profile data.

read-rays
Read a ray file.

read-settings
Read and set boundary conditions from a specified file.

read-surface-clusters
Read surface clusters from a file.

read-transient-table
Read table of transient boundary profile data.

read-viewfactors
Read view factors from a file.

replace-mesh
Replace the mesh with a new one while preserving settings.

set-batch-options
Sets the batch options.

show-configuration
Display current release and version information.

solution-files/
Enter the solution files menu.

- **delete-solution**
  Delete solution files.

- **load-solution**
  Load a solution file.

- **print-solution-files**
  Print a list of available solution files.

start-journal
Start recording all input in a file.

start-transcript
Start recording input and output in a file.

stop-journal
Stop recording input and close journal file.

stop-macro
Stop recording input to a macro.

stop-transcript
Stop recording input and output and close transcript file.

transient-export/

- **abaqus**
  Write an ABAQUS file.

- **ascii**
  Write an ASCII file.

- **avs**
  Write an AVS UCD file.

- **cfd-post-compatible**
  Write data files that are compatible with CFD-Post (i.e., `.cdat` and `.cst` files).

- **cgns**
  Write a CGNS file.

- **dx**
  Write an IBM Data Explorer format file.

- **ensight-gold-transient**
  Write EnSight Gold geometry, velocity, and scalar files.

- **ensight-gold-from-existing-files**
  Write EnSight Gold files using ANSYS FLUENT case files.

`fast`
Write a FAST/Plot3D unstructured mesh velocity scalar file.

`fast-solution`
Write a FAST/Plot3D unstructured solution file.

`fieldview-unstruct`
Write a FIELDVIEW unstructured combined file.

`fieldview-unstruct-mesh`
Write a FIELDVIEW unstructured mesh only file.

`fieldview-unstruct-data`
Write a FIELDVIEW unstructured results only file.

`ideas`
Write an I-deas universal file.

`mechanical-apdl-input`
Write a Mechanical APDL input file.

`nastran`
Write a NASTRAN file.

`patran-neutral`
Write a PATRAN neutral file.

`radtherm`
Write a RadTherm file.

`particle-history-data`
Set up an automatic particle-history data export.

`edit`
Edit transient exports.

`delete`
Delete transient exports.

`settings/`
Enter the automatic export settings menu.

`cfd-post-compatible`
Specify when case files are written with the `.cdat` and `.cst` files exported for ANSYS CFD-Post. Note that this setting is ignored if the **Write Case File Every Time** option is enabled in the **Automatic Export** dialog box.

`write-boundary-mesh`
Write the boundary mesh to a file.

`write-case`
Write a case file.

`write-case-data`
Write a case and a data file.

`write-cleanup-script`
Write the cleanup-script-file for ANSYS FLUENT.

`write-data`
Write a data file.

`write-fan-profile`
Compute radial profiles for a fan zone and write them to a profile file.

write-field-functions
Write the currently defined custom field functions to a file.

write-flamelet
Write a flamelet file.

write-injections
Write out selected DPM injections to a file.

write-isat-table
Write ISAT Table.

write-macros
Write the currently defined macros to a file.

write-pdat?
Enable/disable the attempt to save pdat files.

write-pdf
Write a pdf file.

write-profile
Write surface data as a boundary profile file.

write-settings
Write out current boundary conditions in use.

write-surface-clusters/
Write the surface clusters to a file.

set-parameters
Set the parameters needed for the view factor calculations.

split-angle
Set the split angle for the clustering algorithm.

write-surface-clusters
Compute and write surface clusters for S2S radiation model.

Chapter 6: mesh/

`check`

Perform various mesh consistency checks and display a report in the console that lists the domain extents, the volume statistics, the face area statistics, and any warnings, as well as details about the various checks and mesh failures (depending on the setting specified for `mesh/check-verbosity`).

`check-verbosity`

Set the level of details that will be added to the mesh check report generated by `mesh/check`. A value of 0 (the default) notes when checks are being performed, but does not list them individually. A value of 1 lists the individual checks as they are performed. A value of 2 lists the individual checks as they are performed, and provides additional details (e.g., the location of the problem, the affected cells).

The `check-verbosity` text command can also be used to set the level of detail displayed in the mesh quality report generated by `mesh/quality`. A value of 0 (the default) or 1 lists the minimum orthogonal quality and the maximum aspect ratio. A value of 2 adds information about the zones that contain the cells with the lowest quality, and additional metrics such as the maximum cell squish index and the minimum expansion ratio.

`mesh-info`
Print zone information size.

`make-hanging-interface`
Create hanging interface between quad and tri zones.

`memory-usage`
Report solver memory use.

`modify-zones/`
Enter the zone modification menu. For a description of the items in this menu, see `define/boundary-conditions/modify-zones`.

`polyhedra/`
Enter the polyhedra menu.

`convert-domain`
Convert the entire domain to polyhedra cells.

`convert-hanging-nodes`
Convert cells with hanging nodes/edges to polyhedra.

`convert-skewed-cells`
Convert skewed cells to polyhedra.

`options/`
Enter the polyhedra options menu.

`preserve-interior-zones`

Enable the preservation of surfaces (i.e., manifold zones of type **interior**) during the conversion of the domain to polyhedra. Note that only those zones with a name that includes the string you specify will be preserved.

`quality`

Display information about the quality of the mesh in the console, including the minimum orthogonal quality and the maximum aspect ratio. The level of detail displayed depends on the setting specified for `mesh/check-verbosity`.

`reorder/`

Reorder domain menu.

`band-width`

Print cell bandwidth.

`reorder-domain`

Reorder cells and faces by reverse Cuthill-McKee algorithm.

`reorder-zones`

Reorder zones by partition, type, and ID.

`repair-improve`

`allow-repair-at-boundaries`

Allow the adjustment of the positions of nodes on boundaries as part of the mesh repairs performed by the `mesh/repair-improve/repair` text command.

`improve-quality`

Improve poor quality cells in the mesh, if possible.

`include-local-polyhedra-conversion-in-repair`

Enable/disable the local conversion of degenerate cells into polyhedra based on skewness criteria as part of the mesh repairs performed by the `mesh/repair-improve/repair` text command.

`repair`

Repair mesh problems identified by the mesh check, if possible. The repairs include fixing cells that have the wrong node order, the wrong face handedness, faces that are small or nonexistent, or very poor quality. Only interior nodes are repositioned by default; boundary nodes may be repositioned if the `mesh/repair-improve/allow-repair-at-boundaries` text command is enabled. Note that highly skewed cells may be converted into polyhedra, depending on whether the `mesh/repair-improve/include-local-polyhedra-conversion-in-repair` text command is enabled.

`repair-face-handedness`

Modify cell centroids to repair meshes that contain left-handed faces without face node order problems.

`repair-face-node-order`

Modify face nodes to repair faces with improper face node order and thus eliminate any resulting left-handed faces.

`repair-periodic`

Modify the mesh to enforce a rotational angle or translational distance for periodic boundaries. For translationally periodic boundaries, the command computes an average translation distance and adjusts the node coordinates on the shadow face zone to match this distance. For rotationally periodic boundaries, the command prompts for an angle and adjusts the node coordinates on the shadow face zone using this angle and the defined rotational axis for the cell zone.

`repair-wall-distance`

Correct wall distance at very high aspect ratio hexahedral/polyhedral cells.

`report-poor-elements`
Report invalid and poor quality elements.

`rotate`
Rotate the mesh.

`scale`
Prompt for the scaling factors in each of the active Cartesian coordinate directions.

`size-info`
Print mesh size.

`smooth-mesh`

Smooth the mesh using quality-based, Laplacian, or skewness methods.

`surface-mesh/`
Enter the Surface Mesh menu.

`delete`
Delete surface mesh.

`display`
Display surface meshes.

`read`
Read surface meshes.

`swap-mesh-faces`
Swap mesh faces.

`translate`
Prompt for the translation offset in each of the active Cartesian coordinate directions.

Chapter 7: parallel/

`bandwidth`
Show network bandwidth.

`latency`
Show network latency.

`load-balance`
Enter the load balancing parameters menu.

`physical-models`
Use physical-models load balancing?

`dynamic-mesh`
Use load balancing for dynamic mesh?

`mesh-adaption`
Use load balancing for mesh adaption?

`network/`
Enter the network configuration menu.

`kill-all-nodes`
Delete all compute nodes from virtual machine.

`kill-node`
Kill a specified compute node. The compute node is specified by its integer ID. Compute node 0 can only be killed if it is the last remaining compute node process.

`load-hosts`
Input a hosts database file.

`path`
Specify the path to the `v140/fluent` installation directory. For most cases, the path should never have to be set.

`save-hosts`
Write a hosts file containing all entries in the **Available Hosts** list.

`spawn-node`
Creates a compute node process. It prompts for a hostname and username. If no hostname is specified, the process will be spawned on the spawning machine. If no username is specified, the username of the spawning process will be used.

`partition/`
Enter the partition domain menu.

`auto/`
Set auto partition parameters.

`across-zones`
Enable auto partitioning by zone or by domain.

`load-vector`
Set the auto partition load vector.

method
Set the partition method.

pre-test
Set auto partition pre-testing optimization.

use-case-file-method
Use partitions in a prepartitioned case file.

combine-partition
Merge every N partitions.

merge-clusters
Calls the optimizer that attempts to decrease the number of interfaces by eliminating orphan cell clusters. (An orphan cluster is a group of connected cells such that each member has at least one face that is part of an interface boundary.)

method
Set the partition method.

print-active-partitions
Print active partition information (parallel solver).

print-partitions
Print partition information (serial solver).

print-stored-partitions
Print stored partition information (parallel solver).

reorder-partitions
Reorder partitions.

reorder-partitions-to-architecture
Reorder partitions to architecture.

set/
Enter the set partition parameters menu.

across-zones
Allow partitions to cross zone boundaries (the default). If turned off, it will restrict partitioning to within each cell zone. This is recommended *only* when cells in different zones require significantly different amounts of computation during the solution phase; for example, if the domain contains both solid and fluid zones.

all-off
Turn off all optimizations.

all-on
Turn on all optimizations.

cell-function
Set cell function.

face-area-as-weights
Use face area as connection weights.

isat-weight
Set ISAT weight.

layering
For cases when you want to extrude the partition from specific face zones, this method partitions the cells attached to the selected face zones first, then extrudes the partitions to the other cells (available for serial partitioning only).

load-distribution

Set the number of cells desired for each partition. This is useful, for example, when computing on multiple machines with significantly different performance characteristics. If left unset, each partition will contain an approximately equal number of cells. Normalized relative values may be used for the entries.

merge

Toggle the optimizer that attempts to decrease the number of interfaces by eliminating orphan cell clusters.

nfaces-as-weights

Use number of faces as weights.

origin

Set the x, y, and z coordinate of the origin used by those partitioning functions that require a radial distance. By default, the origin is set to (0, 0, 0).

particle-weight

Set DPM particle weight.

pre-test

Enable the operation that determines the best coordinate-splitting direction.

solid-thread-weight

Use solid thread weights.

smooth

Toggle the optimizer that attempts to minimize the number of interfaces by modifying the partition boundaries to reduce surface area.

stretched-mesh-enhancement

For cases with highly stretched cells, this method improves partition quality by taking cell geometry information into consideration during partitioning with Metis (available for serial partitioning only).

verbosity

Control the amount d tof information that is printed out during partitioning. If set to 1 (the default), a text character `.` is displayed during each bisection, and partition statistics are displayed once the partitioning completes. If set to 2, additional information about the bisection operation is displayed during each bisection. If set to 0, partition statistics and information during each bisection are not displayed.

vof-free-surface-weight

Set VOF free surface weight.

smooth-partition

Call the optimizer that attempts to minimize the number of interfaces by modifying the partition boundaries to reduce surface area.

use-stored-partitions

Use this partitioning.

set/

Enter the set parallel parameters menu.

fast-i/o?

Use the fast I/O option.

partition-mask

Set partition mask.

time-out
Set spawn time-out in seconds.

verbosity
Set the parallel verbosity.

show-connectivity
Print the network connectivity for the selected compute node.

thread-number-control
Set the maximum number of threads on each machine.

timer/
Enter the timer menu.

usage
Print performance statistics in the console window.

reset
Reset domain timers.

Chapter 8: plot/

`circum-avg-axial`
Compute iso-axial band surfaces and plot data vs. axial coordinate on them.

`circum-avg-radial`
Compute iso-radial band surfaces and plot data vs. radius on them.

`change-fft-ref-pressure`
Change reference acoustic pressure.

`fft`
Plot FFT of file data.

`file`
Plot data from an external file.

`file-list`
Plot data from multiple external files.

`file-set/`
Set file plot parameters.

`auto-scale?`
Set the range for the x and y axes. If auto-scaling is not activated for a particular axis, you are prompted for the minimum and maximum data values.

`background-color`
Set the color of the field within the abscissa and ordinate axes.

`key`
Enable/disable display of curve key and set its window title.

`file-lines`
Set parameters for plot lines.

`file-markers`
Set parameters for data markers.

`labels`
Set labels for plot axes.

`lines`
Set parameters for plot lines.

`log?`
Use log scales for one or both axes.

`markers`
Set parameters for data markers.

`numbers`
Set number formats for axes.

`plot-to-file`
Specify a file in which to write XY plot data.

rules
Set parameters for display of major and minor rules.

windows/
XY plot window options. For a description of the items in this menu, see `display/set/windows/xy`.

flamelet-curves/
Enter the flamelet curves menu.

plot-curves
Plot of a curve property.

write-to-file?
Write curve to a file instead of plot.

histogram
Plot a histogram of the specified solution variable using the defined range and number of intervals.

histogram-set/
Set histogram plot parameters. Sub-menu items are the same as `file-set/` above.

plot
Plot solution on surfaces.

plot-direction
Set plot direction for XY plot.

residuals
Contains commands that allow you to select the variables for which you want to display XY plots of residual histories in the active graphics window.

residuals-set/
Set residual plot parameters. Sub-menu items are the same as `file-set/` above.

solution
Plot solution on surfaces and/or zones. Zone and surface names can be indicated using a wildcard (*).

solution-set/
Set solution plot parameters. Sub-menu items are the same as `file-set/` above.

label-alignment
Set the alignment of the xy plot label to be horizontal or axis aligned.

Chapter 9: report/

dpm-histogram/
Enter the DPM histogram menu.

compute-sample
Compute the minimum/maximum of a sample variable.

delete-sample
Delete a sample from the loaded sample list.

list-samples
Show all samples in a loaded sample list.

plot-sample
Plot a histogram of a loaded sample.

read-sample
Read a sample file and add it to the sample list.

set/
Enter the settings menu for the histogram.

auto-range?
Automatically compute the range of the sampling variable for histogram plots.

correlation?
Compute the correlation of the sampling variable with another variable.

cumulation-curve?
Compute a cumulative curve for the sampling variable or correlation variable when `correlation?` is specified.

diameter-statistics?
Compute the Rosin Rammler parameters, Sauter, and other mean diameters.

histogram-mode?
Use bars for the histogram plot or xy-style.

maximum
Specify the maximum value of the x-axis variable for histogram plots.

minimum
Specify the minimum value of the x-axis variable for histogram plots.

number-of-bins
Specify the number of bins.

percentage?
Use percentages of bins to be computed.

variable^3?
Use the cubic of the cumulation variable during computation of the cumulative curve.

weighting?
Use weighting with additional variables when sorting data into samples.

`write-sample`
Write a histogram of a loaded sample into a file.

`dpm-sample`
Sample trajectories at boundaries and lines/planes.

`dpm-summary`
Print discrete phase summary report.

`element-mass-flow`
Print list of element flow rate at inlets and outlets. This reports the mass flow rates of all chemical elements (in kg/s) flowing through the simulation boundaries.

`fluxes/`
Enter the fluxes menu.

`heat-transfer`
Print heat transfer rate at boundaries.

`heat-transfer-sensible`
Print the sensible heat transfer rate at the boundaries.

`mass-flow`
Print mass flow rate at inlets and outlets.

`rad-heat-trans`
Print radiation heat transfer rate at boundaries.

`forces/`
Enter the forces menu.

`pressure-center`
Print the center of pressure on wall zones.

`wall-forces`
Compute the forces along the specified force vector for all wall zones.

`wall-moments`
Compute the moments about the specified moment center for all wall zones.

`heat-exchanger/`
Enter the heat exchanger menu.

`computed-heat-rejection`
Print total heat rejection.

`inlet-temperature`
Print inlet temperature.

`outlet-temperature`
Print outlet temperature.

`mass-flow-rate`
Print mass flow rate.

`specific-heat`
Print fluid's specific heat.

`particle-summary`
Print summary report for all current particles.

`path-line-summary`
Print pathline summary report.

print-histogram
Print a histogram of a scalar quantity.

projected-surface-area
Compute the area of the projection of selected surfaces along the x, y, or z axis.

reference-values/
Enter the reference value menu.

area
Set reference area for normalization.

compute/
Compute reference values from zone boundary conditions.

density
Set reference density for normalization.

depth
Set reference depth for volume calculation.

enthalpy
Set reference enthalpy for enthalpy damping and normalization.

length
Set reference length for normalization.

list
List current reference values.

pressure
Set reference pressure for normalization.

temperature
Set reference temperature for normalization.

velocity
Set reference velocity for normalization.

viscosity
Set reference viscosity for normalization.

zone
Set reference zone.

species-mass-flow
Print list of species mass flow rate at inlets and outlets. This reports the mass flow rates of all species (in kg/s) flowing through the simulation boundaries.

summary
Print the current settings for physical models, boundary conditions, material properties, and solution parameters.

surface-integrals/
Enter the surface integral menu.

area
Print the area of the selected surfaces.

area-weighted-average
Print area-weighted average of the specified quantity over the selected surfaces.

facet-avg
Print the facet average of the specified quantity over the selected surfaces.

`facet-max`
Print the maximum of the specified quantity over facet centroids of the selected surfaces.

`facet-min`
Print the minimum of the specified quantity over facet centroids of the selected surfaces.

`flow-rate`
Print the flow rate of the specified quantity over the selected surfaces.

`integral`
Print the integral of the specified quantity over the selected surfaces. You can include a wildcard (*) within the surface names.

`mass-flow-rate`
Print the mass flow rate through the selected surfaces.

`mass-weighted-avg`
Print the mass-averaged quantity over the selected surfaces.

`standard-deviation`
Print the standard deviation of the scalar at the facet centroids of the surface.

`sum`
Print sum of scalar at facet centroids of the surfaces.

`uniformity-index-area-weighted`
Print the area-weighted uniformity index of the specified quantity over the selected surfaces.

`uniformity-index-mass-weighted`
Print the mass-weighted uniformity index of the specified quantity over the selected surfaces.

`vertex-avg`
Print the vertex average of the specified quantity over the selected surfaces.

`vertex-max`
Print the maximum of the specified quantity over vertices of the selected surfaces.

`vertex-min`
Print the minimum of the specified quantity over vertices of the selected surfaces.

`volume-flow-rate`
Print the volume flow rate through the selected surfaces.

`system/`
Enter the system menu.

`proc-stats`
Print ANSYS FLUENT process information. This is used to report the status of each of the ANSYS FLUENT processes, including memory and CPU usage.

`sys-stats`
System information. This is used to report the status of the machines where ANSYS FLUENT processes have been spawned, including memory and CPU status.

`uds-flow`
Print list of user-defined scalar flow rate at boundaries.

`volume-integrals/`
Enter the volume integral menu.

`mass-avg`
Print mass-average of scalar over cell zones.

mass-integral
Print mass-weighted integral of scalar over cell zones.

maximum
Print maximum of scalar over all cell zones.

minimum
Print minimum of scalar over all cell zones.

sum
Print sum of scalar over all cell zones.

volume
Print total volume of specified cell zones.

volume-avg
Print volume-weighted average of scalar over cell zones.

volume-integral
Print integral of scalar over cell zones.

Chapter 10: solve/

`animate/`
Enter the animation menu.

`define/`
Enter the animation definition menu.

`define-monitor`
Define new animation.

`edit-monitor`
Change animation monitor attributes.

`playback/`
Enter the animation playback menu.

`delete`
Delete animation sequence.

`play`
Play the selected animation.

`read`
Read new animation from file or already-defined animations.

`write`
Write animation sequence to the file.

`dpm-update`
Update discrete phase source terms.

`dual-time-iterate`
Perform unsteady iterations for a specified number of time steps.

`execute-commands/`
Enter the execute commands menu.

`add-edit`
Add or edit execute commands.

`disable`
Disable an execute command.

`enable`
Enable an execute command.

`initialize/`
Enter the flow initialization menu.

`compute-defaults/`
Enter the compute default values menu.

`axis`
Compute flow-initialization defaults from a zone of this type.

`all-zones`
Initialize the flow field with the default values.

`zone`
You can select the type of zone from which you want to compute these values. The types of zones available are:

- `exhaust-fan`
- `fan`
- `fluid`
- `inlet-vent`
- `intake-fan`
- `interface`
- `interior`
- `mass-flow-inlet`
- `network`
- `network-end`
- `outflow`
- `outlet-vent`
- `periodic`
- `porous-jump`
- `pressure-far-field`
- `pressure-inlet`
- `pressure-outlet`
- `radiator`
- `rans-les-interface`
- `recirculation-inlet`
- `recirculation-outlet`
- `shadow`
- `solid`
- `symmetry`
- `velocity-inlet`
- `wall`

`dpm-reset`
Reset discrete phase source terms to zero.

`fmg-initialization`
Initialize using the full-multigrid initialization (FMG).

`hyb-initialization`
Initialize using the hybrid initialization method.

`init-flow-statistics`
Initialize unsteady statistics.

initialize-flow
Initialize the flow field with the current default values.

init-instantaneous-vel
Initialize unsteady velocity.

list-defaults
List default values.

open-channel-auto-init
Open channel automatic initialization.

reference-frame
Set reference frame to absolute or relative.

repair-wall-distance
Correct wall distance at very high aspect ratio hexahedral/polyhedral cells.

set-defaults/
Set default initial values.

set-fmg-initialization/
Enter the set full-multigrid for initialization menu. Initial values for each variable can be set within this menu.

set-hyb-initialization/
Enter the hybrid initialization menu.

general-settings
Enter the general-settings menu.

turbulence-settings
Enter the turbulence-settings menu.

species-settings
Enter the species-settings menu.

show-time-sampled
Display the amount of simulated time covered by the data sampled for unsteady statistics.

iterate
Perform a specified number of iterations.

Note

This option is still available during transient simulations, since it can be used to add more iterations to the same time step after interrupting iterations within a time step.

mesh-motion
Perform mesh motion.

monitors/
Set solution monitors.

force/
Enter the force monitors menu.

clear-monitors
Discard the internal and external file data associated with specified or all force monitors.

delete-monitors
Delete a specified monitor, so that it is not available for future simulations.

list-monitors
Print the details of all of the created monitors in the console.

monitor-unsteady-iters?
Specify (for transient calculations) whether the monitors are updated every iteration or every time step.

set-drag-monitor
Set the parameters for a new or existing drag coefficient monitor, including the list of wall zones on which to compute the coefficient, whether to print, plot, and/or write the data, the name of the output file (if appropriate), the plot window, and the force vector associated with the coefficient.

set-lift-monitor
Set the parameters for a new or existing lift coefficient monitor, including the list of wall zones on which to compute the coefficient, whether to print, plot, and/or write the data, the name of the output file (if appropriate), the plot window, and the force vector associated with the coefficient.

set-moment-monitor
Set the parameters for a new or existing moment coefficient monitor, including the list of wall zones on which to compute the coefficient, whether to print, plot, and/or write the data, the name of the output file (if appropriate), the plot window, and the moment center and moment vector associated with the coefficient.

residual/
Enter the residual monitors menu.

check-convergence?
Choose which currently-monitored residuals should be checked for convergence.

convergence-criteria
Set convergence criteria for residuals that are currently being both monitored and checked.

criterion-type
Set convergence criterion type.

monitor?
Choose which residuals to monitor as printed and/or plotted output.

n-display
Set the number of most recent residuals to display in plots.

n-maximize-norms
Set the number of iterations through which normalization factors will be maximized.

normalization-factors
Set normalization factors for currently-monitored residuals (if `normalize?` is set to `yes`).

normalize?
Choose whether to normalize residuals in printed and plotted output.

n-save
Set number of residuals to be saved with data. History is automatically compacted when buffer becomes full.

plot?
Choose whether residuals will be plotted during iteration.

print?
Choose whether residuals will be printed during iteration.

relative-conv-criteria
Set relative convergence criteria for residuals that are currently being both monitored and checked.

re-normalize
Re-normalize residuals by maximum values.

reset?
Choose whether to delete the residual history and reset iteration counter to 1.

scale-by-coefficient?
Choose whether to scale residuals by coefficient sum in printed and plotted output.

window
Specify window in which residuals will be plotted during iteration.

statistic/
Enter the statistic monitors menu.

monitors
Choose which statistics to monitor as printed and/or plotted output.

plot?
Choose whether or not statistics will be plotted during iteration.

print?
Choose whether or not statistics will be printed during iteration.

window
Specify first window in which statistics will be plotted during iteration. Multiple statistics are plotted in separate windows, beginning with this one.

surface/
Contains commands to control surface monitoring.

clear-data
Clear current surface monitor data.

clear-monitors
Remove all defined surface monitors.

curves/
Enter the curves menu.

lines
Set lines parameters for surface monitors.

markers
Set markers parameters for surface monitors.

list-monitors
List defined surface monitors.

set-monitor
Define or modify a surface monitor.

volume/
Contains commands to control volume monitoring.

clear-data
Clear current volume monitor data.

clear-monitors
Remove all defined volume monitors.

list-monitors
List defined volume monitors.

set-monitor
Define or modify a volume monitor.

patch
Patch a value for a flow variable in the domain.

set/
Enter the set solution parameters menu.

adaptive-time-stepping
Set adaptive time stepping parameters.

bc-pressure-extrapolations
Set pressure extrapolations schemes on boundaries.

If you are using the density-based solver, you will be asked the following questions:

extrapolate total quantities on pressure-outlet boundaries?
The default is `[no]`. If you enter `yes`, and the flow leaving the pressure outlet is subsonic, then the total pressure and total temperature from the domain's interior are extrapolated to the boundary and used with the imposed static pressure to determine the full thermodynamic state at the boundary.

extrapolate pressure on pressure-inlet boundary?
The default is `[no]`. If you enter `yes`, then for cases with very low Mach number flow in the single-precision density-based solver, you can improve convergence by using pressure extrapolation instead of the default velocity extrapolation scheme.

pressure on pressure-outlet b.c. is obtained via an advection splitting method?
The default is `[yes]`. If you choose the default, this means that the pressure-outlet boundary condition implementation in the density-based solver has an absorption behavior, as described in Calculation Procedure at Pressure Outlet Boundaries of the User's Guide. To revert to pre-ANSYS FLUENT 6.3 boundary condition implementations, where the pressure on the faces of a pressure-outlet boundary is fixed to the specified value while the flow is subsonic, enter `no`.

Important

The absorption behavior of the pressure-outlet boundary condition should not be confused with rigorous non-reflecting boundary condition implementation, described in Non-Reflecting Boundary Conditions of the User's Guide.

If you are using the pressure-based solver, you will be asked the following questions:

extrapolate pressure on flow inlets?
The default is `[yes]`.

extrapolate pressure on all boundaries?
The default is `[no]`.

extrapolate velocity on out-flow boundaries?
The default is `[no]`.

convergence-acceleration-for-stretched-meshes/
Enable convergence acceleration for stretched meshes to improve the convergence of the implicit density based solver on meshes with high cell stretching.

correction-tolerance/
Enter the correction tolerance menu.

coupled-vof-expert
Set coupled vof expert controls. You will be prompted with the following questions:

Use linearized buoyancy force?
provides the implicit linearization of buoyancy force.

Use blended treatment for buoyancy force?
will turn off buoyancy linearization in certain unstable conditions.

Use false time step linearization?
provides additional stability for buoyancy driven flows in the steady state pseudo-transient mode by increasing the diagonal dominance using false time step size.

Use smoothed density for pseudo-transient method?
smooths the cell density near the interface, thus avoiding unphysical acceleration of lighter phase in the vicinity of interface. This option is only available for steady state pseudo-transient method.

Note

There is an additional entry for the number of density smoothings (default 2), which can be increased in case of very large unphysical velocities across the interface.

courant-number
Set the fine-grid Courant number (time step factor). This command is available only for the coupled solvers.

data-sampling
Enable data sampling for unsteady flow statistics.

disable-reconstruction?
Completely disables reconstruction, resulting in totally first-order accuracy.

discretization-scheme/
Enter the discretization scheme menu. This allows you to select the discretization scheme for the convection terms in the solution equations.

pressure
Select which Pressure model is to be used. Five models are available:

Index	Model
10	Standard
11	Linear
12	Second Order
13	Body Force Weighted
14	PRESTO!

mp
Select which convective discretization scheme for volume fraction is to be used. Six models are available:

Index	Model
0	First Order
1	Second Order
28	Compressive
5	Modified HRIC
29	BGM
4	QUICK

mom

Select which Momentum model is to be used. Five models are available:

Index	Model
0	First Order Upwind
1	Second Order Upwind
2	Power Law
4	QUICK
5	Third-Order MUSCL

The Energy and Turbulence models are indexed as in the Momentum model table above.

Contact your ANSYS FLUENT technical support engineer for more details.

equations/

Select the equations to be solved.

expert

Set expert options.

extrapolate-eqn-vars/

Enter the extrapolation menu.

extrapolate-vars?

Applies a predictor algorithm for computing initial conditions at time step n+1.

flow-warnings?

Specify whether or not to print warning messages when reversed flow occurs at inlets and outlets, and when mass flow inlets develop supersonic regions. By default, flow warnings are printed.

flux-type

Set the flux type.

gradient-scheme

Set gradient options.

heterogeneous-stiff-chemistry

Set the heterogeneous stiff-chemistry solver.

high-order-term-relaxation/

Enter the High Order Term Relaxation menu.

enable?

Enable/Disable High Order Term Relaxation.

options/

High Order Term Relaxation Options.

relaxation-factor
Set the relaxation factor.

variables/
Select the variables.

limiter-warnings?
Specify whether or not to print warning messages when quantities are being limited. By default, limiter warnings are printed.

limits
Set solver limits for various solution variables, in order to improve the stability of the solution.

mp-mfluid-aniso-drag
Set anisotropic drag parameters for the Eulerian multiphase model.

mp-reference-density
Set the reference density option for the Eulerian multiphase model. The following options are available:

Index	VOF Equation Discretization	Option
0	mass conservative	reference density for a particular phase in a cell is treated as the volume averaged density of that phase in the whole domain
1	mass conservative	reference density for a particular phase in a cell is treated as the density of that phase in that cell
2	mass conservative	reference density for any phase in a cell is treated as the mixture density of that phase in that cell
3	volume conservative	reference density for a particular phase in a cell is treated as the density of that phase in that cell

max-corrections/
Enter the set max-corrections menu.

max-flow-time
Set the maximum flow time.

max-iterations-per-time-step
Set the number of time steps for a transient simulation.

Note

This option is available when automatic initialization and case modification is enabled.

multi-grid-amg
Set the parameters that govern the algebraic multigrid procedure.

multi-grid-controls/
Set multigrid parameters and termination criteria.

multi-grid-fas
Set the parameters that control the FAS multigrid solver. This command appears only when the explicit coupled solver is used.

multi-stage
Set the multi-stage coefficients and the dissipation and viscous evaluation stages. This command appears only when the explicit coupled solver is used.

number-of-iterations
Set the number of iterations for a steady state simulation without starting the calculation.

number-of-time-steps
Set the number of time steps for a transient simulation without starting the calculation.

numerical-beach-control
Set damping function in flow direction. This command appears only when the VOF model is enabled. Select the damping function to be used:

Index	Damping Function
0	Linear
1	Quadratic
2	Cubic
3	Cosine

numerics
Set numerics options.

open-channel-inlet-controls
Set Froude number update frequency for steady state and volume fraction interpolation method at inlet for sub-critical flow. This command appears only when the VOF model is enabled.

p-v-controls
Set pressure-velocity controls.

p-v-coupling

Select which pressure-velocity coupling model is to be used. Four models are available:

Index	Model
20	SIMPLE
21	SIMPLEC
22	PISO
24	Coupled

phase-based-vof-discretization
Set phase based slope limiter for VOF compressive scheme.

poor-mesh-numerics
Solution correction on meshes of low quality.

poor-mesh-numerics-quality-based?
Enable/disable poor mesh numerics on cells with low quality.

predict-next-time?
Applies a predictor algorithm for computing.

pseudo-transient-expert/
Enter the pseudo transient expert usage control menu.

pseudo-relaxation-factor/
Enter the pseudo relaxation factor menu.

pseudo-transient
Set the pseudo transient formulation.

reactions?
Enable the species reaction sources and set relaxation factor.

relaxation-factor/
Enter the relaxation-factor menu.

relaxation-method
Set the solver relaxation method.

reporting-interval
Set the number of iterations for which convergence monitors are reported. The default is 1 (after every iteration).

residual-smoothing
Set the implicit residual smoothing parameters. This command is available only for the explicit coupled solver.

residual-tolerance/
Enter the residual tolerance menu.

residual-verbosity
Set the amount of residual information to be printed. A value of 0 (the default) prints residuals at the end of each fine grid iteration. A value of 1 prints residuals after every stage of the fine grid iteration. A value of 2 prints residuals after every stage on every grid level.

set-all-species-together
Set all species discretizations and URFs together.

set-controls-to-default
Set controls to default values.

set-solution-steering
Set solution steering parameters.

slope-limiter-set/
Select a new Fluent solver slope limiter.

solution-steering
Enable solution steering for the density-based solver.

stiff-chemistry
Set solver options for stiff chemistry solutions.

surface-tension
Set surface-tension calculation options.

time-step
Set the magnitude of the (physical) time step Δt.

unsteady-statistics-cff
Unsteady statistics for custom field functions.

under-relaxation/
Enter the under-relaxation menu, which allows you to set the under-relaxation factor for each equation that is being solved in a segregated manner.

undo-timestep
When enabled, if the truncation error within a time step exceeds the specified tolerance FLUENT will automatically undo the current calculation and make another attempt with the time step reduced by 1/2. This will be attempted up to 5 times after which FLUENT will accept the result and proceed to the next time step.

`variable-time-stepping`
Set variable time-stepping options for VOF explicit schemes.

`vof-numerics`
Set VOF numeric options.

`update-physical-time`
Advance the unsteady solution to the next physical time level. Using this command in conjunction with the `iterate` command allows you to manually advance the solution in time (rather than doing it automatically with the `dual-time-iterate` command).

Chapter 11: surface/

`circle-slice`
Extract a circular slice.

`delete-surface`
Remove a defined data surface.

`iso-clip`
Clip a data surface (surface, curve, or point) between two isovalues.

`iso-surface`
Extract an iso-surface (surface, curve, or point) from the current data field.

`line-slice`
Extract a linear slice in 2D, given the normal to the line and a distance from the origin.

`line-surface`
Define a “line" surface by specifying the two endpoint coordinates.

`list-surfaces`
Display the ID and name, and the number of point, curve, and surface facets of the current surfaces.

`mouse-line`
Extract a line surface that you define by using the mouse to select the endpoints.

`mouse-plane`
Extract a planar surface defined by selecting three points with the mouse.

`mouse-rake`
Extract a “rake" surface that you define by using the mouse to select the endpoints.

`partition-surface`
Define a data surface consisting of mesh faces on the partition boundary.

`plane`
Create a plane given 3 points bounded by the domain.

`plane-bounded`
Create a bounded surface.

`plane-point-n-normal`
Create a plane from a point and normal.

`plane-slice`
Extract a planar slice.

`plane-surf-aligned`
Create a plane aligned to a surface.

`plane-view-plane-align`
Create a plane aligned to a view-plane.

`point-array`
Extract a rectangular array of data points.

`point-surface`
Define a “point" surface by specifying the coordinates.

`quadric-slice`
Extract a quadric slice.

`rake-surface`
Extract a “rake" surface, given the coordinates of the endpoints.

`rename-surface`
Rename a defined data surface.

`sphere-slice`
Extract a spherical slice.

`surface-cells`
Extract all cells intersected by a data surface.

`transform-surface`
Transform surface.

`zone-surface`
Create a surface of a designated zone and gives it a specified name.

Chapter 12: turbo/

2d-contours
Display 2D contours.

avg-contours
Display average contours.

compute-report
Compute turbomachinery quantities.

current-topology
Set the current turbo topology for global use.

write-report
Write the turbo report to file.

xy-plot-avg
Display average XY plots.

Chapter 13: views/

auto-scale
Scale and center the current scene without changing its orientation.

camera/
Enter the camera menu to modify the current viewing parameters.

dolly-camera
Adjust the camera position and target.

field
Set the field of view (width and height).

orbit-camera
Adjust the camera position without modifying the target.

pan-camera
Adjust the camera target without modifying the position.

position
Set the camera position.

projection
Toggles between perspective and orthographic views.

roll-camera
Adjust the camera up-vector.

target
Set the point to be the center of the camera view.

up-vector
Set the camera up-vector.

zoom-camera
Adjust the camera's field of view. This operation is similar to dollying the camera in or out of the scene. Dollying causes objects in front to move past you. Zooming changes the perspective effect in the scene (and can be disconcerting).

default-view
Reset view to front and center.

delete-view
Remove a view from the list.

last-view
Return to the camera position before the last manipulation.

list-views
List predefined and saved views.

read-views
Read views from a view file.

`restore-view`
Use a saved view.

`save-view`
Save the current view to the view list.

`write-views`
Write selected views to a view file.

Appendix A. Text Command List Changes in ANSYS FLUENT 14.0

For a complete listing of changes to the Text Command List for ANSYS FLUENT 14.0, refer to "Text Command List and Settings Changes" in the FLUENT Migration Manual.