

FLUENT 6.3 Text Command List

September 2006

Copyright © 2006 by Fluent Inc.
All Rights Reserved. No part of this document may be reproduced or otherwise used in
any form without express written permission from Fluent Inc.

Airpak, FIDAP, FLUENT, FLUENT for CATIA V5, FloWizard, GAMBIT, Icemax, Icepak,
Icepro, Icewave, Icechip, MixSim, and POLYFLOW are registered trademarks of Fluent
Inc. All other products or name brands are trademarks of their respective holders.

CHEMKIN is a registered trademark of Reaction Design Inc.

Portions of this program include material copyrighted by PathScale Corporation
2003-2004.

Fluent Inc.
Centerra Resource Park
10 Cavendish Court
Lebanon, NH 03766

Contents

adapt/	1
define/	5
display/	25
exit	37
file/	39
grid/	45
parallel/	47
plot/	51
report/	53
solve/	57
surface/	63
turbo/	65
view/	67

adapt/

adapt-to-gradients Adapt grid 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 of refinement level differences.

adapt-to-register Adapt grid based on the selected adaption register and adaption limits.

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.

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 rectangular 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-max-volume** Mark cells for adaption based on maximum allowed volume.
- 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-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.

refine-bndry-cells Refines the boundary cells based on the specified list of zones and adaption limits.

refine-large-cells Refine cells that are larger than prescribed volume.

refine-to-vol-change Refine cells with large changes in cell volume.

set/ Enter the adaption set menu.

- cell-zones** Set cell zones to be used for marking adaption.
- coarsen-grid?** Turn on/off ability to coarsen grid.
- 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.
- hanging-node?** Turn on/off the use of hanging node adaption. When this option is turned off, the conformal adaption method will be used.
- init-node-flags** Initialize the node flags.
- max-number-cells** Limit the total number of cells produced by refinement.
- min-cell-volume** Restrict the size of the cells considered for refinement.
- orig-node-settings** Prompt for the desired node removal status of original boundary, internal, and periodic nodes.
- reconstruct-geometry** Enable/disable geometry-based adaption.
- refine-grid?** Turn on/off grid adaption by point addition.
- refn-node-settings** Prompt for the desired node removal status of boundary, internal, and periodic nodes produced by adaptive grid refinement.
- set-geometry-controls** Set geometry controls for wall zones.

smooth-grid Smooth the grid using Laplace or skewness methods.

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

boundary-conditions/ Enter the boundary conditions menu.

axis Set boundary conditions for a zone of this type.

copy-bc Copy boundary conditions to other zones. To copy conditions to all zones of a certain type, use a * in the name (e.g., specify **wall-*** as the name of the zone to which you want to copy the conditions).

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 description of the **Boundary Conditions** panel.

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

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

modify-zone/ Enter the modify zones menu.

activate-cell-zone Activate cell thread.

append-mesh Append new mesh.

append-mesh-data Append new mesh with data.

deactivate-cell-zone Deactivate cell thread.

create-all-shell-threads Create all shells.

delete-all-shells Delete 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 grid.
- orient-face-zone** Orient the face zone.
- repair-face-handedness** Reverse orientation of left-handed faces.
- repair-periodic** Modify the grid 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.
- 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 info. (default), 2 : detailed info. for debugging.
- show-status** Show current status of non-reflecting b.c.'s.

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 FLUENT not supported in FLUENT

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-method** Select method for setting the mass flow rate.
- verbosity?** Enable/disable verbosity when using targeted mass flow rate.

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-zones/ Enter the dynamic zone boundary conditions menu.

create Create dynamic zone.

delete Delete dynamic zone.

insert-boundary-layer Insert new cell zone.

insert-interior-layer Insert new layer cell zone at specified location.

list List dynamic zones.

remove-boundary-layer Remove cell zone.

remove-interior-layer Remove interior layer cell zone.

grid-interfaces/ Enter the grid-interfaces menu.

create Create a grid-interface.

delete Delete a grid-interface.

draw Draw specified sliding interface zone.

list List all grid-interfaces.

make-periodic Make interface zones periodic.

recreate Recreate all currently defined grid interfaces.

reset Delete all sliding-interfaces.

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

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.

delete Delete a material from local storage.

list-materials List all locally-stored materials.

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

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.

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.

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.

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

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

`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.

`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.

cloud-max-diameter Set the maximum cloud diameter.

cloud-min-diameter Set the minimum cloud diameter.

cloud-on? Enable/disable cloud tracking.

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.

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

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

-
- `tracking-scheme` Specify a tracking scheme.
- `error-control?` Adapt integration step length based on a maximum error.
- `automated-scheme-selection?` Enable/disable the adaptation of integration step length based on a maximum error.
- `drag-law` Set the drag law. on particle temperature.
- `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.
- `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.
- `parallel/` Enter the parallel menu to set parameters for parallel DPM calculations.
- `enable-workpile?` Turn on/off particle workpile algorithm.
- `n-threads` Set the number of processors to use for DPM.
- `report` Print particle workpile statistics.
- `use-message-passing` Set DPM parallel-mode to message passing.
- `use-shared-memory` Set DPM parallel-mode to shared memory.

spray-model/ Enter the spray model menu.

droplet-collision? Enable/disable droplet collision model.

droplet-breakup? Enable/disable spray breakup model.

number-of-particles-per-cell Set the average number of particles per cell.

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

unsteady-tracking Enable/disable unsteady particle tracking.

user-defined Set DPM user-defined functions.

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

dynamic-mesh-controls/ Enter the dynamic mesh control menu.

auto-hide-cells? Enable/disable automatic hiding of skewed cells.

events/ Enter the dynamic mesh events menu.

export-event-file Export dynamic mesh events to file.

import-event-file Import dynamic mesh event file.

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

crank-angle-step Specify crank angle step size.

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

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

minimum-stroke Specify cut off point for in-cylinder piston.

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

piston-data Specify piston stroke and connecting rod length.

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-parameter/ 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-parameter/ Enter the dynamic mesh remeshing menu.

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

face-remeshing? Enable/disable local face remeshing at deforming zones.

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.

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-parameter/ 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 spring-based smoothing in tri/tet cell zones.

smoothing-parameter/ Enter the dynamic mesh smoothing menu.

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

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

`constant-factor` Set the spring constant relaxation factor.

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

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

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

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

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

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

`energy?` Enable/disable the energy model.

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

`multiphase/` Enter the multiphase model menu.

`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 NO_x model.

`nox-parameters/` Enter the NO_x parameters menu.

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

`nox-chemistry` Select NO_x chemistry model.

`nox-turbulence-interaction` Set NO_x turbulence interaction model.

radiation/ Enter the radiation models menu.

- discrete-ordinates?** Enable/disable discrete ordinates radiation model.
- discrete-transfer?** Enable/disable discrete transfer radiation model.
- do-coupling?** Enable/disable DO/energy coupling.
- dtrm-parameters/** Enter the dtrm parameters menu.
 - check-ray-file** Read DTRM rays file.
 - controls** Set dtrm solution controls.
 - make-globs** Make globs (coarser grid) for radiation.
 - ray-trace** Create DTRM rays for radiation.
- 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-vf-only** Compute/write view factors only.
 - compute-write-vf** Compute/write surface clusters and view factors for S2S radiation model.
 - enable-poly-to-poly-vf** Compute view factors directly for polygon to polygon surface radiation.
 - partial-enclosure-temperature** Set temperature for the partial enclosure.
 - 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 view factor 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.

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

`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.

`soot-model-parameters` Select soot model parameters.

`soot-process-parameters` Select soot process parameters.

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

`sox?` Enable/disable the SOx model.

`sox-parameters/` Enter the SOx parameters menu.

- `inlet-diffusion?` Enable/disable inclusion of diffusion at inlets.
- `sox-chemistry` Select the SOx chemistry model.
- `sox-turbulence-interaction` Set the SOx/turbulence interaction model.

`species/` Enter the species models menu.

- `clear-isat-table` Kill ISAT table.
- `diffusion-energy-source?` Enable/disable diffusion energy source.
- `flamelet-expert` Set flamelet expert parameters.
- `heat-of-surface-reactions?` Enable/disable heat of surface reactions.
- `ignition-model?` Enable/disable the ignition model.
- `ignition-model-controls` Set ignition model parameters.
- `inlet-diffusion?` Enable/disable inclusion of diffusion at inlets.
- `integration-parameters` Set ISAT parameters.
- `KINetics-from-Reaction-Design?` Enable/disable KINetics from Reaction Design.
- `KINetics-parameters/` Enter the expert KINetics 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.
- `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.

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-combustion? Enable/disable premixed combustion 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.

thermal-diffusion? Enable/disable thermal diffusion.

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? 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.

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-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-omega turbulence model.

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

`kw-transitional?` Enable/disable the k - ω transitional flow option.

`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.

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

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

- `ke-multiphase-models` Select k - ϵ multiphase model.
- `multiphase-options` Enable/disable multiphase options.
- `rsm-multiphase-models` Select Reynolds Stress 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.
- `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.

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

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

`kato-launders-model?` Enable/disable Kato-Launders 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	Launders-Sharma
3	Yang-Shih
4	Abe-Kondoh-Nagano
5	Chang-Hsieh-Chen

Contact your Fluent technical support engineer for more details.

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

i 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.

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

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

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

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

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

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

`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.

`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.

`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.

turbo/ Enter the turbo menu.

define-topology Define a turbo topology.

grid-method Set turbo structured grid 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.

heat-exchanger/ Enter the heat exchanger 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.

interpreted-functions Load interpreted user-defined functions.

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

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

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

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.

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 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.

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

`grid-outline` Display the grid boundaries.

`grid-partition-boundary` Display grid partition boundaries.

`hard-copy` Generate a hardcopy of the active window.

`multigrid-coarsening` Display a coarse grid 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.

`profile` Display profiles of a flow variable.

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

`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 grids by type or by ID.

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

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

`grid-inlet` Set the color of inlet faces.

`grid-interior` Set the color of interior faces.

`grid-internal` Set the color of internal interface faces.

`grid-outlet` Set the color of outlet faces.

`grid-periodic` Set the color of periodic faces.

`grid-symmetry` Set the color of symmetric faces.

`grid-axis` Set the color of axisymmetric faces.

- `grid-free-surface` Set the color of free-surface faces.
 - `grid-traction` Set the color of traction faces.
 - `grid-wall` Set the color of wall faces.
 - `grid-interface` Set the color of grid interfaces.
 - `list` List available colors.
 - `reset-colors` Reset individual grid surface colors to the defaults.
 - `skip-label` Set the number of labels to be skipped in the colopmap 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-grid?` Choose whether to render grid on top of contours, vectors, etc.
 - `surfaces` Set the surfaces on which contours are drawn.
- `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-grid?` Determine whether the grids are drawn as wireframe or solid.
- `grid-level` Set coarse grid level to be drawn.
- `grid-partitions?` Turn on/off option to draw grid partition boundaries.
- `grid-surfaces` Set surface IDs to be drawn as grids.
- `grid-zones` Set zone IDs to be drawn as grids.
- `hard-copy/` Enter the hardcopy options menu.
- `color-mode/` Enter the hardcopy 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.

- 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.
 - hpgl** Produce HPGL output for hardcopies.
 - image** Produce SGI IRIS image output for hardcopies.
 - jpeg** Produce JPEG output for hardcopies.
 - 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. (This is the default file type.)
 - 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.
 - 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.

-
- 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.
 - sphere-attrib** Specify the size and number of slices to be used in drawing spheres.
 - 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).
- 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.
 - 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-grid?** Enable/disable rendering the grid 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.

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

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 **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.

`wireframe-animation?` Enable/disable use of wireframes during animation.

`reset-graphics` Reset the graphics system.

`title` Set problem title.

`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-grid?` Choose whether to render grid 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.

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.

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.

-
- `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.
 - `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 the `surface/` chapter.
 - `surface-cells` Draw the cells on the specified surfaces.
 - `surface-grid` Draw the grid defined by the specified surfaces.
 - `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 the `view/` chapter.
 - `zone-grid` Draw the grid defined by specified face zones.

exit

exit Exit program.

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.

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

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

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

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

confirm-overwrite? Confirm attempts to overwrite existing files.

define-macro Save input to a named macro.

export/ Export case and data information.

abaqus Write an ABAQUS file.

ansys Write an ANSYS file.

ansys-input Write an ANSYS input file.

ascii Write an ASCII file.

avs Write an AVS UCD file.

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-grid` Write FAST/Plot3D unstructured grid 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-grid` Write FIELDVIEW unstructured grid-only file.

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

`fieldview-unstruct` Write FIELDVIEW unstructured combined file.

`gambit` Write GAMBIT neutral file.

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

`ideas` Write an I-deas universal file.

`nastran` Write a NASTRAN file.

`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.

`execute-macro` Run a previously defined macro.

`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.

`ansys/` Import an ANSYS file.

- `input` Read an ANSYS file as a case file.
- `result` Read an ANSYS result 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.

`fluent4-case` Import a formatted 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.

`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 a FLUENT case file during read.

- `metis` Read and partition a FLUENT case file.
- `metis-zone` Read and partition a 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.
- `result` Read a PATRAN result file as a case file.

`plot3d/` Import a PLOT3D file.

- `grid` Read a PLOT3D file as a case file.
- `result` Read a PLOT3D result 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-bc` Read and set boundary conditions from specified file.

`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-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-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.

`set-batch-options` Sets the batch options.

`show-configuration` Display current release and version information.

`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.

`write-bc` Write out current boundary conditions in use.

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

`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.

`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 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-isat-table` Write ISAT Table.

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

`write-pdf` Write a pdf file.

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

`check` Perform various grid consistency checks.

`grid-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-zone`.

`polyhedra/` Enter the polyhedra menu.

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

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

`options/` Enter the polyhedra options menu.

`parallel-migration-by-file` Enable cell migration by file input/output during `convert-domain`.

`quality` Analyze the quality of the grid.

`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.

`rotate` Rotate the grid.

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

`size-info` Print grid size.

`smooth-grid` Smooth the grid using Laplace or skewness methods.

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

`delete` Delete surface mesh.

`display` Display surface meshes.

`read` Read surface meshes.

`swap-grid-faces` Swap grid faces.

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

parallel/

load-balancing Set load balancing parameters.

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 **Fluent.Inc** 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.

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.

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.

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).

pre-test Enable the operation that determines the best coordinate-splitting direction.

smooth Toggle the optimizer that attempts to minimize the number of interfaces by modifying the partition boundaries to reduce surface area.

verbosity Control the amount of 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.

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.

`timer/` Enter the timer menu.

`usage` Print performance statistics in the console window.

`reset` Reset domain timers.

parallel/

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.

fft Plot FFT of file data.

change-fft-ref-pressure Change reference acoustic pressure.

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.

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.

write-to-file? Write curve to a file instead of plot.

plot-curves Plot of a curve property.

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.

solution-set/ Set solution plot parameters. Sub-menu items are the same as **file-set/** above.

`dpm-sample` Sample trajectories at boundaries and lines/planes.

`dpm-summary` Print discrete phase summary report.

`fluxes/` Enter the fluxes menu.

`heat-transfer` Print heat transfer rate at 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.

`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.

 - 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.
 - 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.
 - 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.

 - 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.

`vol-avg` Print volume-weighted average of scalar over cell zones.

`vol-integral` Print integral of scalar over cell zones.

`volume` Print total volume of specified cell zones.

`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. You can select the type of zone from which you want to compute these values.

`dpm-reset` Reset discrete phase source terms to zero.

`fmg-initialization` Initialize using the full-multigrid initialization (FMG).

`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.

`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.

iterate Perform a specified number of iterations.

mesh-motion Perform mesh motion.

monitors/ Set solution monitors.

force/ Enter the force monitors menu.

clear-all-monitors-data Discard the internal and external file data associated with the force monitors.

clear-drag-monitor-data Discard the internal and external file data (if any exist) for the drag coefficient.

clear-lift-monitor-data Discard the internal and external file data (if any exist) for the lift coefficient.

clear-moment-monitor-data Discard the internal and external file data (if any exist) for the moment coefficient.

drag-coefficient Set the parameters for monitoring the drag coefficient, 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.

lift-coefficient Set the parameters for monitoring the lift coefficient, 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.

moment-coefficient Set the parameters for monitoring the moment coefficient, 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, the moment center associated with the coefficient, and which component of the moment vector to monitor.

monitor-unsteady-iters? Specify (for transient calculations) whether the monitors are updated every iteration or every time step.

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.

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

`export-particle-data` Export particle history data.

`import-particle-data` Import particle history data.

`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.

`correction-tolerance/` Enter the correction tolerance menu.

`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, as well as the pressure-velocity coupling scheme for the pressure-based solver.

`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!

flow Select which Pressure-Velocity Coupling model is to be used. Three models are available:

Index	Model
20	SIMPLE
21	SIMPLEC
22	PISO

mom Select which Momentum model is to be used. Four models are available:

Index	Model
0	First Order Upwind
1	Second Order Upwind
2	Power Law
4	QUICK

The Energy and Turbulence models are indexed as in the Momentum model table above.

Contact your Fluent technical support engineer for more details.

equations/ Select the equations to be solved.

expert Set expert options.

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.

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.

max-corrections/ Enter the set max-corrections menu.

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.

numerics Set numerics options.

predict-next-time? Applies a predictor algorithm for computing.

-
- `p-v-controls` Set pressure-velocity controls.
- `p-v-coupling` Select the pressure-velocity coupling scheme.
- `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-controls-to-default` Set controls to default values.
- `slope-limiter-set/` Select a new Fluent solver slope limiter.
- `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 .
- `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.
- `variable-time-stepping` Set variable time-stepping options for VOF explicit schemes.
- `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).

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 grid 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.

`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.

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.

`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 camera position before 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.