Copyright and Trademark Information

© 2013 SAS IP, Inc. All rights reserved. Unauthorized use, distribution or duplication is prohibited.

ANSYS, ANSYS Workbench, Ansoft, AUTODYN, EKM, Engineering Knowledge Manager, CFX, FLUENT, HFSS and any and all ANSYS, Inc. brand, product, service and feature names, logos and slogans are registered trademarks or trademarks of ANSYS, Inc. or its subsidiaries in the United States or other countries. ICEM CFD is a trademark used by ANSYS, Inc. under license. CFX is a trademark of Sony Corporation in Japan. All other brand, product, service and feature names or trademarks are the property of their respective owners.

Disclaimer Notice

THIS ANSYS SOFTWARE PRODUCT AND PROGRAM DOCUMENTATION INCLUDE TRADE SECRETS AND ARE CONFIDENTIAL AND PROPRIETARY PRODUCTS OF ANSYS, INC., ITS SUBSIDIARIES, OR LICENSORS. The software products and documentation are furnished by ANSYS, Inc., its subsidiaries, or affiliates under a software license agreement that contains provisions concerning non-disclosure, copying, length and nature of use, compliance with exporting laws, warranties, disclaimers, limitations of liability, and remedies, and other provisions. The software products and documentation may be used, disclosed, transferred, or copied only in accordance with the terms and conditions of that software license agreement.

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

U.S. Government Rights

For U.S. Government users, except as specifically granted by the ANSYS, Inc. software license agreement, the use, duplication, or disclosure by the United States Government is subject to restrictions stated in the ANSYS, Inc. software license agreement and FAR 12.212 (for non-DOD licenses).

Third-Party Software

See the legal information in the product help files for the complete Legal Notice for ANSYS proprietary software and third-party software. If you are unable to access the Legal Notice, please contact ANSYS, Inc.

Published in the U.S.A.
## Table of Contents

1. adapt/ ................................................... ................................................... ............................................. 1  
2. define/ ................................................... ................................................... ........................................... 5  
3. display/ ................................................... ................................................... ....................................... 51  
4. exit / close-fluent ................................................... ................................................... ............... 71  
5. file/ .................................................................................................................. 73  
6. mesh/ .............................................................................................................. 85  
7. parallel/ ......................................................................................................... 89  
8. plot/ ............................................................................................................... 93  
9. report/ ............................................................................................................ 95  
10. solve/ ........................................................................................................... 101  
11. surface/ ....................................................................................................... 115  
12. switch-to-meshing-mode .............................................................................. 117  
13. turbo/ ........................................................................................................... 119  
14. views/ ........................................................................................................... 121  
A. Text Command List Changes in ANSYS Fluent 15.0 ................................................... 123
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 adaptation 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.

copy-bc
Copy boundary conditions to other zones.

degassing
Set boundary conditions for a zone of this type.

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.

change-zone-phase
Set the phase (liquid or vapor) for a specific fluid zone.

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 Fluent User’s Guide) or set up a non-conformal interface (see Using a Non-Conformal Mesh in ANSYS Fluent in the Fluent 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, for example, 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 that 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.

**slit-interior-between-diff-solids**
Slit the interior zone between solid zones of differing materials to create a coupled wall. You will generally be prompted by Fluent if this is necessary.

**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. The use of asterisks (*) as wildcards is supported.

**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).

**verbosity**
Enable/disable nrbc verbosity scheme output.

**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.

open-channel-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.

phase-shift/
  Enter the phase shift settings menu.

  multi-disturbances
    Set basic phase-shift parameters.

  extra-settings
    Set other phase-shift parameters.

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.

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. The use of asterisks (*) as wildcards is supported.

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 (that is, 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.

contact-parameters/
Enter the dynamic mesh contact-parameters menu. This text command is only available when you enable contact detection using the prompts of the define/dynamic-mesh/dynamic-mesh? text command.

contact-threshold
Specify threshold distance for contact detection.

contact-udf
Select the UDF to be invoked when contact is detected.

contact-zones
Select face zones involved in contact detection.

flow-control?
Enable/disable flow control.

flow-control-parameters/
Set up flow control zones

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 (that is, 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
The boundary node relaxation is used by spring smoothing. The boundary node relaxation allows you to relax the update of the node positions at deforming boundaries. A value of 0
prevents deforming boundary nodes from moving and a value of 1 indicates no under-relaxation.

**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.

**diffusion-fvm?**
Answering yes at the prompt changes the diffusion-based smoothing method to the cell-based finite volume approach that was the default in releases prior to Fluent 15.0. Answering no at the prompt changes the diffusion-based smoothing method to the default node-based finite element method.

**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 the smoothing method (spring, diffusion, or linearly elastic solid) used by the dynamic mesh model.

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

**spring-on-simplex-shapes?**
Enable/disable spring-based smoothing for triangular / tetrahedral cells in mixed element zones.
poisson-ratio
   Set the Poisson’s ratio used by the linearly elastic solid model.

verbosity
   Setting this to 1 will cause smoothing residuals to be printed to the text console. The default value of 0 suppresses this output.

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.

transient-settings/
   Enter the transient dynamic mesh settings menu. This text command is only available when you enable dynamic mesh using the prompts of the define/dynamic-mesh/dynamic-mesh? text command. Solver time must also be set to Transient.

   allow-second-order?
       Enable/disable second order transient scheme for dynamic mesh cases.

verbosity
   Enable/disable transient scheme verbosity for dynamic mesh cases.

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` or the `define/mesh-morpher-optimizer/region/define-bounding-box` 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.

**read-scaling-factors-from-file**
Read the scaling factor settings from an ASCII text file.

**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**
Assign scaling factors and a single parameter to a single control point in a region.

**set-parameters-multiple**
Assign scaling factors and multiple parameters to multiple control points in a region.

**write-scaling-factors-from-file**
Write the scaling factor settings to an ASCII text file.

**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` or the `define/mesh-morpher-optimizer/region/define-bounding-box` 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` or the `define/mesh-morpher-optimizer/region/define-bounding-box` 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. This menu is not available when the NEWUOA optimizer is selected using the `define/mesh-morpher-optimizer/optimizer-parameters/optimizer-type` text command.

**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.

disable-mesh-check
 Specify whether you want to disable the general mesh check that is part of the optimization process. This check is conducted immediately after the mesh is deformed in every design stage, and determines whether a solution is calculated. Disabling this check allows you to use mesh repair commands (which can be set up using the define/mesh-morpher-optimizer/op-timizer-parameters/initial-commands text command) at the start of a design stage, so that an accurate solution can be calculated.

disable-mesh-check
 Specify whether you want to disable the general mesh check that is part of the optimization process. This check is conducted immediately after the mesh is deformed in every design stage, and determines whether a solution is calculated. Disabling this check allows you to use mesh repair commands (which can be set up using the define/mesh-morpher-optimizer/op-timizer-parameters/initial-commands text command) at the start of a design stage, so that an accurate solution can be calculated.

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. This text command is not available when the NEWUOA optimizer is selected using the define/mesh-morpher-optimizer/optimizer-parameters/optimizer-type text command.

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. This text command is not available when the NEWUOA optimizer is selected using the define/mesh-morpher-optimizer/optimizer-parameters/optimizer-type text command.

initialization
 Specify how the solution variables should be treated after the mesh is deformed, that is, whether they should be initialized to the values defined in the Solution Initialization task page, remain the values obtained in the previous design iteration, or be read from a data file that you specify.

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.

mesh-quality-check
 Specify if orthogonal quality should be used to determine whether a solution is calculated for a mesh, and define the minimum orthogonal quality value allowed (values may range from 0–1, where 0 represents the worst quality).

monitor/
 Enter the monitor menu in order to plot and/or record optimization history data, that is, how the value of the objective function varies with each design stage produced by the mesh morpher/optimizer. This text menu is not available when the NEWUOA optimizer is selected using the define/mesh-morpher-optimizer/optimizer-parameters/optimizer-type text command.

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.

newuoa-initial-parameter-variation
Define how much the parameters will be allowed to vary during the initial calculations. This text command is only available when the NEWUOA optimizer is selected using the define/mesh-morpher-optimizer/optimizer-parameters/optimizer-type text command.

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. The objective-function-definition text command is not available when the NEWUOA optimizer is selected using the define/mesh-morpher-optimizer/optimizer-parameters/optimizer-type text command.

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 to be used. You can select one of the six built-in optimizers (1–5 and 7), or specify that you will use Design Exploration in ANSYS Workbench (6). Note that 6 is only available if you have launched your ANSYS Fluent session from ANSYS Workbench. For information about how the built-in optimizers function or how to use Design Exploration, see Modeling Flows Using the Mesh Morph/Optimizer in the User’s Guide or Working With Input and Output Parameters in Workbench in the ANSYS Fluent in Workbench User’s Guide, respectively.

save-case-data-files
Set up the automatic saving of intermediate case and data files during the optimization run, by specifying: the frequency (in design iterations) with which file sets are saved; the maximum number of file sets retained (after the maximum limit has been saved, the earliest file set will be overwritten with the latest); and the root name assigned to the files (which will have the design iteration number appended to it).

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

parameter-settings/
Enter the parameters menu.

number-of-parameters
Define the number of parameters available to be assigned to control points.
**parameter-bounds**
Define the minimum and maximum values allowed by the built-in optimizer for the parameters. This text command is only available if the define/mesh-morpher-optimizer/optimizer? text command is enabled.

**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. Note that you can enter a numeric value or define an input parameter (if you have enabled the definition of input parameters using the define/parameters/enable-in-TUI? text command). The parameter-value text command is only available in the following situations:

- if the define/mesh-morpher-optimizer/optimizer? text command is disabled
- if the define/mesh-morpher-optimizer/optimizer? text command is enabled and 6 is selected for the optimizer type via the define/mesh-morpher-optimizer/optimizer-type text command

**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 by specifying the name, number of control points, dimensions, origin coordinates, and the components of the direction vectors. 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.

**define-bounding-box**
Create a new deformation region by specifying the name, the bounding zones that best represent the extents of the deformation region, and the number of control points. 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.

**scaling-enlarge**
Set the scaling factor applied to the bounding box when you click the Enlarge button in the Regions tab of the Mesh Morpher/Optimizer dialog box.

**scaling-reduce**
Set the scaling factor applied to the bounding box when you click the Reduce button in the Regions tab of the Mesh Morpher/Optimizer dialog box.

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

**create**
Create a mixing plane.
**define/**

**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.
define/

dpm/
   Enter the 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.

fill-injection-material-sources
   Initialize the DPM sources corresponding to each material.
injections/
Enter the injections menu.

create-injection
Create an injection.

delete-injection
Delete an injection.

injection-properties/
Enter the menu to set or display properties for one or more injections.

list/
Enter the menu to display the current properties for one or more injections.

list-picked-injections
List the injections that have been selected for display using pick-injections-to-list.

list-picked-properties-to-list
List the properties that have been selected for display using pick-properties-to-list.

list-property-values
List the settings for the properties selected with pick-properties-to-list. Only available when a single injection has been selected with pick-injections-to-list.

list-uniform-values
List the settings for properties selected with pick-properties-to-list that have uniform values for all of the injections selected with pick-injections-to-list.

list-values-per-injection
For each injection selected with pick-injections-to-list, list the values for properties selected with pick-properties-to-list.

list-values-per-property
For each property selected with pick-properties-to-list, list the values for the injections selected with pick-injections-to-list.

pick-injections-to-list
Select the injection or injections for which property values will be displayed. The use of asterisks (*) as wildcards is supported.

pick-properties-to-list
Select the property or properties for which values will be displayed. The use of asterisks (*) as wildcards is supported.

set/
Enter the menu to set physical models such as drag and breakup for one or more injections.
**list-picked-injections**
List the injections selected for setting with `pick-injections-to-set`.

**physical-models**
Enter the menu to set physical models such as drag and breakup for the selected injections.

**brownian-motion**
Enable Brownian motion effects for the currently selected injection(s).

**drag-parameters**
Set the drag law and corresponding parameters for the currently selected injection(s).

**spray-secondary-breakup**
Enter the menu for setting the breakup model and parameters for the currently selected injection(s). Available commands are as those described under `define/models/dpm/spray-model/` with the addition of the following.

**no-breakup**
Disable breakup for the selected injection(s) only.

**pick-injections-to-set**
Select the injection or injections for which properties will be set. The use of asterisks (*) as wildcards is supported.

**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.

**Important**
Drag and breakup model parameters for each injection are set instead in `/define/models/dpm/injections/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.

**linearized-dpm-source-terms?**
Enable/disable linearization of source terms for the discrete phase.

**no.-of-cont-phase-its-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.

**average-DDPM-variables?**
Enable/disable mesh node averaging of DDPM quantities.

**average-each-step?**
Enable/disable mesh node averaging during integration time step.

**average-kernel**
Specify the averaging kernel to use for mesh node averaging.

**average-source-terms?**
Enable/disable mesh node averaging of DPM source terms.
coupled-heat-mass-update
   Enable/disable coupled heat and mass update.

drag-law
   Set the drag law.

enable-node-based-averaging?
   Enable/disable mesh node averaging of DPM quantities.

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

gaussian-factor
   Specify the Gaussian constant when using the gaussian kernel for mesh node averaging.

minimum-liquid-fraction
   A droplet evaporates completely when the remaining mass is below this fraction of the initial droplet mass.

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.

enable-contour-plots
   Enable computation of mean and/or RMS values of additional discrete phase variables for postprocessing.

ensemble-average
   Ensemble average cloud properties.

erosion-accretion
   Enable/disable erosion/accretion.

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

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

maximum-udf-species
   Specify the maximum number of species that will be accessible from discrete phase model UDFs. Only species with indices up to this value are accessible in discrete phase model UDFs.
**particle-radiation**
   Enable/disable particle radiation.

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

**pressure-gradient-force**
   Enable/disable inclusion of pressure gradient effects in the particle force balance.

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

**stagger-radius**
   Specify the region over which to spatially stagger particles when particle-staggering is enabled for non-atomizer injections.

**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.

**uniform-mass-distribution-for-injections?**
   Specify a uniform distribution of mass over the cross section of solid cone and atomizer injections. This can become important when the mesh is smaller than the diameter (or another characteristic size) of the injection.

**vaporization-options**
   Set Vaporization options.

**virtual-mass-force**
   Enable/disable inclusion of the virtual mass force in the particle force balance.

**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.

**hybrid-2domain?**
   Enables or disables the use of a second domain for DPM particle tracking.

**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-coalescence?
Enable/disable droplet coalescence when using the stochastic collision model.

droplet-collision?
Enable/disable droplet collision model.

enable-breakup?
Enable/disable breakup globally, but do not alter injection settings other than enable/disable.

khrt-model
Enable KHRT breakup model.

set-breakup
Enable/disable breakup model globally and uniformly specify injection breakup parameters.

ssd-model
Enable SSD breakup model.

tab-model
Enable TAB breakup model.

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

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

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.

denergy?
   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.
onoptions
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, view factors, 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.

shell-conduction/
Enter the shell conduction models menu.

multi-layer-shell?
Enable/disable the ability to define multi-layer shell conduction for walls.

read-csv
Define the shell conduction settings by reading a CSV file.
settings
   Enable shell conduction and define the settings for any wall or group of walls by manually entering the number and properties of the layers.

write-csv
   Write your saved shell conduction settings to a CSV file.

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.

c 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 chemistry ODE integrator parameters. Enable/disable stiff chemistry acceleration methods and set their 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.

**partially-premixed-properties**
Set/change partially-premixed mixture properties. This command is only available when partially-premixed-combustion? is enabled.

**re-cac1-par-premix-props**
Re-calculate partially-premixed properties. This command is only available when partially-premixed-combustion? is enabled.

**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.

**reactor-network-model?**
Enable/disable the Reactor Network Model.

**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
Switch between the R15 and R14.5 spark models and 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.

add-intermittency-transition-model?
Enable/disable the Intermittency Transition model to account for transitional effects. This text command is only available for the SST $k$-$\omega$, Scale-Adaptive Simulation with SST, and Detached Eddy Simulation with SST models.

buoyancy-effects?
Enable/disable effects of buoyancy on turbulence.
curvature-correction?
  Enable/disable the curvature correction.

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

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

inviscid?
  Enable/disable inviscid flow model.

ke-easm?
  Enable/disable the EASM $k-\varepsilon$ turbulence model.

ke-realizable?
  Enable/disable the realizable $k-\varepsilon$ turbulence model.

ke-rng?
  Enable/disable the RNG $k-\varepsilon$ turbulence model.

ke-standard?
  Enable/disable the standard $k-\varepsilon$ turbulence model.

k-kl-w?
  Enable/disable the $k$-kl-$\omega$ turbulence model.

kw-easm?
  Enable/disable the EASM $k$-$\omega$ turbulence model.

kw-low-re-correction?
  Enable/disable the $k$-$\omega$ low Re option.

kw-shear-correction?
  Enable/disable the $k$-$\omega$ shear-flow correction option.

kw-sst?
  Enable/disable the SST $k$-$\omega$ turbulence model.

kw-standard?
  Enable/disable the standard $k$-$\omega$ turbulence model.

laminar?
  Enable/disable laminar flow model.

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

les-subgrid-dynamic-fvar?
  Enable/disable the dynamic subgrid-scale mixture fraction variance 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-s_minus_omega?
   Enable/disable the WMLES S-Ω 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 FLUENT 14.

**sas?**
Enable/disable Scale-Adaptive Simulation (SAS) in combination with the SST $k-\omega$ 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.
turb-compressibility?
Enable/disable the compressibility correction 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.

production-limiter?
Enable/disable Production Limiter modification.

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 $k-\varepsilon$ turbulence model.

low-re-ke-index
Specify which low-Reynolds-number $k-\varepsilon$ model is to be used. Six models are available:

<table>
<thead>
<tr>
<th>Index</th>
<th>Model</th>
</tr>
</thead>
<tbody>
<tr>
<td>0</td>
<td>Abid</td>
</tr>
<tr>
<td>1</td>
<td>Lam-Bremhorst</td>
</tr>
<tr>
<td>2</td>
<td>Launder-Sharma</td>
</tr>
<tr>
<td>3</td>
<td>Yang-Shih</td>
</tr>
<tr>
<td>4</td>
<td>Abe-Kondoh-Nagano</td>
</tr>
<tr>
<td>5</td>
<td>Chang-Hsieh-Chen</td>
</tr>
</tbody>
</table>

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 $C_\mu$ definition for the realizable $k-\varepsilon$ model.

**Important**

Note that the use of the realizable $k-\varepsilon$ 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.

kw-add-des?
Enable/disable Detached Eddy Simulation (DES) in combination with the currently selected transition SST turbulence model. This text command is only available for transient cases.

kw-add-sas?
Enable/disable Scale-Adaptive Simulation (SAS) in combination with the currently selected $\omega$-based URANS turbulence model. This text command is only available for transient cases.

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.

set-phase
Select phase for real gas EOS subcritical condition.
**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 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 the cpp utility included with the ANSYS Fluent installation.

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.

display/

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.

reacting-channel-curves
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 must 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/**

Define 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-line-method**

Specify the method to perform hidden line rendering. This command will appear only when `hidden-lines?` is true.

- **normal-hlr-algorithm** - Normal hidden lines algorithm. This is the default.
- **mesh-display-hlr?** - For removing hidden lines for surfaces that are very close together. Use this if `normal-hlr-algorithm` is not working. This will only work for meshes.

**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 mesh 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.

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

logo-color
   Set logo color to white/black in graphics 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. (for example, %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.

text/
  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.

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-list-tree-separator
    Set the separator character for list tree.

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. 115).

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. 121).

zone-mesh

Draw the mesh defined by specified face zones. Zone names can be indicated using wildcards (*).
Chapter 4: \texttt{exit / close-fluent}

\texttt{exit}

Exit program.

\texttt{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.

close-without-save?
Exits ANSYS Fluent without saving data in Workbench. This command is only available when running ANSYS Fluent in Workbench.

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.

**surface-energy-loss**
Maps the total surface loss (that is, 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.

**volumetric-energy-loss**
Maps the total volumetric loss (that is, 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 (that is, .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 (that is, .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 ICEM CFD 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.
Write a NASTRAN file.

Export particle-history data.

Write a PATRAN neutral file.

Write a PATRAN nodal results file.

Export RADTHERM file.

Write a Tecplot+3DV format file.

Enter the fluid-structure interaction menu.

Display the mesh for a fluid-structure interaction.

Read an FEM mesh for one-way data mapping from ANSYS Fluent.

Write a fluid-structure interaction mesh file.

Import case and data information.

Import an ABAQUS file.

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

Read an ABAQUS input file as a case file.

Read an ABAQUS odb file as a case file.

Import a CFX file.

Read a CFX definition file as a case file.

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.

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.

reload-setup
  Discards any changes preformed in the current ANSYS Fluent in Workbench session and removes any corresponding data from the Solution cell. This command is only available when running ANSYS Fluent in Workbench.

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.

dsolution-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.

sync-workbench
  Directly update Workbench with the most recent Fluent changes. This command is only available when running ANSYS Fluent in Workbench.

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 (that is, .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 (for example, 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 (that is, 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`.

**redistribute-boundary-layer**
Redistribute the nodes in a boundary layer zone to achieve a desired growth rate after anisotropic adaptation.

**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 therefore 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: \texttt{parallel/}

\textbf{bandwidth}\par Show network bandwidth.

\textbf{gpgpu/}\par Enter the GPGPU menu

\begin{description}
\item[select] Select which GPGPUs to use for AMG acceleration
\item[show] list the available GPGPUs. GPGPUs selected for use are indicated by the presence of an asterisk (*).
\end{description}

\textbf{latency}\par Show network latency.

\textbf{load-balance}\par Enter the load balancing parameters menu.

\begin{description}
\item[physical-models] Use physical-models load balancing?
\item[dynamic-mesh] Use load balancing for dynamic mesh?
\item[mesh-adaption] Use load balancing for mesh adaption?
\end{description}

\textbf{network/}\par Enter the network configuration menu.

\textbf{kill-all-nodes}\par Delete all compute nodes from virtual machine.

\textbf{kill-node}\par 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.

\textbf{load-hosts}\par Input a hosts database file.

\textbf{path}\par Specify the path to the \texttt{v150/fluent} installation directory. For most cases, the path should never have to be set.

\textbf{save-hosts}\par Write a hosts file containing all entries in the \textbf{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 pre-partitioned 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 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.

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.

film-heat-transfer
Print wall film heat transfer rate at boundaries. This text command is only available when you enable the Eulerian wall film model.

film-mass-flow
Print wall film mass flow rate at boundaries. This text command is only available when you enable the Eulerian wall film model.

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**
Print total mass of a phase within a selected cell zone.

**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.

twopisum
   Print sum of scalar over all cell zones multiplied by \(2\pi\).

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:

- degassing
- 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.

corvergence/
Enter the convergence menu to add surface, volume, drag, lift and moment monitors to convergence criteria.

add-edit
Add or edit convergence criteria for surface, volume, drag, lift and moment monitors.

condition
Option to stop the calculations. All convergence conditions are met or any convergence condition is met.

delete
Delete a monitor from convergence criteria.

frequency
To set how often convergence checks are done with respect to iterations or time steps.

list
List defined convergence criteria for monitors.

set-average-over
Specify the number of iterations/time steps Fluent calculates the running average over.

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-itors?
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.

amg-options/
Enter the AMG options menu

amg-gpgpu-execution
specify whether to use GPGPU acceleration for coupled and/or scalar systems.

conservative-amg-coarsening?
enable the use of conservative coarsening techniques for coupled equations that can improve parallel performance and/or convergence for some difficult cases.

laplace-coarsening?
enable Laplace coarsening for scalar and/or coupled equations.

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].

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?
smoothes the cell density near the interface, therefore 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:

<table>
<thead>
<tr>
<th>Index</th>
<th>Model</th>
</tr>
</thead>
<tbody>
<tr>
<td>10</td>
<td>Standard</td>
</tr>
<tr>
<td>11</td>
<td>Linear</td>
</tr>
<tr>
<td>12</td>
<td>Second Order</td>
</tr>
<tr>
<td>13</td>
<td>Body Force Weighted</td>
</tr>
<tr>
<td>14</td>
<td>PRESTO!</td>
</tr>
</tbody>
</table>

mp
Select which convective discretization scheme for volume fraction is to be used. Six models are available:

<table>
<thead>
<tr>
<th>Index</th>
<th>Model</th>
</tr>
</thead>
<tbody>
<tr>
<td>0</td>
<td>First Order</td>
</tr>
<tr>
<td>1</td>
<td>Second Order</td>
</tr>
<tr>
<td>28</td>
<td>Compressive</td>
</tr>
<tr>
<td>5</td>
<td>Modified HRIC</td>
</tr>
<tr>
<td>29</td>
<td>BGM</td>
</tr>
<tr>
<td>4</td>
<td>QUICK</td>
</tr>
</tbody>
</table>

mom
Select which Momentum model is to be used. Five models are available:
<table>
<thead>
<tr>
<th>Index</th>
<th>Model</th>
</tr>
</thead>
<tbody>
<tr>
<td>0</td>
<td>First Order Upwind</td>
</tr>
<tr>
<td>1</td>
<td>Second Order Upwind</td>
</tr>
<tr>
<td>2</td>
<td>Power Law</td>
</tr>
<tr>
<td>4</td>
<td>QUICK</td>
</tr>
<tr>
<td>5</td>
<td>Third-Order MUSCL</td>
</tr>
</tbody>
</table>

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.

lock-solid-temperature?
Specify whether you want to lock (or “freeze”) the temperature values for all the cells in solid zones (including those to which you have hooked an energy source through a UDF) and in walls that have shell conduction enabled, so that the values do not change during further solver iterations.

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:

<table>
<thead>
<tr>
<th>Index</th>
<th>VOF Equation Discretization</th>
<th>Option</th>
</tr>
</thead>
<tbody>
<tr>
<td>0</td>
<td>mass conservative</td>
<td>reference density for a particular phase in a cell is treated as the volume averaged density of that phase in the whole domain</td>
</tr>
<tr>
<td>1</td>
<td>mass conservative</td>
<td>reference density for a particular phase in a cell is treated as the density of that phase in that cell</td>
</tr>
<tr>
<td>2</td>
<td>mass conservative</td>
<td>reference density for any phase in a cell is treated as the mixture density of that phase in that cell</td>
</tr>
<tr>
<td>3</td>
<td>volume conservative</td>
<td>reference density for a particular phase in a cell is treated as the density of that phase in that cell</td>
</tr>
</tbody>
</table>

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.

**nb-gradient-boundary-option?**
switches between the modified treatment of node-based gradients at boundary cells and the legacy treatment (R14.5.7 and earlier).

**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:

<table>
<thead>
<tr>
<th>Index</th>
<th>Damping Function</th>
</tr>
</thead>
<tbody>
<tr>
<td>0</td>
<td>Linear</td>
</tr>
<tr>
<td>1</td>
<td>Quadratic</td>
</tr>
<tr>
<td>2</td>
<td>Cubic</td>
</tr>
<tr>
<td>3</td>
<td>Cosine</td>
</tr>
</tbody>
</table>

**numerics**
Set numerics options.

**open-channel-controls**
For flows that do not transition from sub-critical to super-critical, or vice-versa, you can speed-up the solution calculation by updating the frequency of Froude number during runtime.

**p-v-controls**
Set pressure-velocity controls.

**p-v-coupling**
Select which pressure-velocity coupling model is to be used. Four models are available:

<table>
<thead>
<tr>
<th>Index</th>
<th>Model</th>
</tr>
</thead>
<tbody>
<tr>
<td>20</td>
<td>SIMPLE</td>
</tr>
<tr>
<td>21</td>
<td>SIMPLEC</td>
</tr>
<tr>
<td>22</td>
<td>PISO</td>
</tr>
<tr>
<td>24</td>
<td>Coupled</td>
</tr>
</tbody>
</table>

**phase-based-vof-discretization**
Set phase based slope limiter for VOF compressive scheme.

**poor-mesh-numerics/**
Enter the poor mesh numerics menu.

**enable?**
Solution correction on meshes of low quality.
cell-quality-based?
Enable/disable poor mesh numerics on cells with low quality.

user-defined-on-register
Include a register for the poor mesh numerics or not.

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 $\Delta t$.

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.

vof-explicit-controls
   Set the sub time step calculation method for VOF calculations.

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: switch-to-meshing-mode

switch-to-meshing-mode

Switch from the solution mode to the meshing mode. This text command is only available if you have not yet read a mesh or a case file.
Chapter 13: 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 14: 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.
views/

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 15.0

For a complete listing of changes to the Text Command List for ANSYS Fluent 15.0, refer to Text Command List and Settings Changes in the Fluent Migration Manual.