



ANSYS Fluent Text Command List



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

Release 15.0
November 2013

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

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 adaption function.

init-node-flags

Initialize the node flags.

max-level-refine

Set maximum level of refine in the mesh.

max-number-cells

Limit the total number of cells produced by refinement.

min-cell-volume

Restrict the size of the cells considered for refinement.

min-number-cells

Set limit on the number of cells in the mesh.

reconstruct-geometry

Enable/disable geometry-based adaption.

refine-mesh?

Turn on/off mesh adaption by point addition.

set-geometry-controls

Set geometry controls for wall zones.

smooth-mesh

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

swap-mesh-faces

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

Chapter 2: define/

boundary-conditions/

Enter the boundary conditions menu.

Important

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

axis

Set boundary conditions for a zone of this type.

bc-settings/

Enter the boundary conditions settings menu.

mass-flow

Select method for setting the mass flow rate.

pressure-outlet

Select pressure specification method on pressure-outlet boundaries.

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.

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

outflow
Set boundary conditions for a zone of this type.

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

periodic
Set boundary conditions for a zone of this type.

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/optimizer-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 Morpher/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.

delete

Delete a mixing plane.

list

List defined mixing plane(s).

set/

Set global parameters relevant to mixing planes.

averaging-method

Set the mixing plane profile averaging method.

under-relaxation

Set mixing plane under-relaxation factor.

fix-pressure-level

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

conserve-swirl/

Enter the menu to set swirl conservation in mixing plane menu.

enable?

Enable/disable swirl conservation in mixing plane.

verbosity?

Enable/disable verbosity in swirl conservation calculations.

report-swirl-integration

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

conserve-total-enthalpy/

Enter the menu to set total enthalpy conservation in mixing plane menu.

enable?

Enable/disable total enthalpy conservation in mixing plane.

verbosity?

Enable/disable verbosity in total-enthalpy conservation calculations.

models/

Enter the models menu to configure the solver.

acoustics/

Enter the acoustics menu.

auto-prune

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

broad-band-noise?

Enable/disable the broadband noise model.

convective-effects?

Enable/disable the convective effects option.

compute-write

Compute sound pressure.

cylindrical-export?

Enable/disable the export of data in cylindrical coordinates.

display-flow-time?

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

export-source-data-cgns?

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

export-volumetric-sources?

Enable/disable the export of fluid zones.

export-volumetric-sources-cgns?

Enable/disable the export of fluid zones.

ffowcs-williams?

Enable/disable the Ffowcs-Williams-and-Hawkings model.

moving-receiver?

Enable/disable the moving receiver option.

off?

Enable/disable the acoustics model.

read-compute-write

Read acoustic source data files and compute sound pressure.

receivers

Set acoustic receivers.

sources

Set acoustic sources.

write-acoustic-signals

Write on-the-fly sound pressure.

write-centroid-info

Write centroid info.

addon-module

Load addon module.

axisymmetric?

Specify whether or not the domain is axisymmetric.

crevice-model?

Enable/disable the crevice model.

crevice-model-controls/

Enter the crevice model controls menu.

dpm/

Enter 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-iters-per-dpm-iter

Set the frequency with which the particle trajectory calculations are introduced.

reset-sources-at-timestep?

Enable/disable flush of DPM source terms at beginning of every time step.

underrelaxation-factor

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

update-dpm-sources-every-flow-iteration?

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

numerics/

Enter the numerics menu to set numerical solution parameters.

automated-scheme-selection?

Enable/disable the adaptation of integration step length based on a maximum error.

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.

energy?
Enable/disable the energy model.

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

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

initialize-wallfilm-model
Initialize Eulerian Wall Film Model.

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

model-options
Set Eulerian Wall Film Model Options.

film-material
Set Film Material and Properties.

solution-options
Set Eulerian Wall Film Model Solution Options.

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

heat-exchanger/
Enter the heat exchanger menu.

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

add-heat-exchanger
Add heat-exchanger.

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

delete-heat-exchanger
Delete heat-exchanger.

heat-exchanger?

Enable/disable the dual cell heat-exchanger model.

modify-heat-exchanger

Modify heat-exchanger.

plot-NTU

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

write-NTU

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

macro-model/

Enter the heat macro-model menu.

delete-heat-exchanger-group

Delete heat-exchanger group.

heat-exchanger?

Enable/disable heat-exchanger model.

heat-exchanger-group

Define heat-exchanger group.

heat-exchanger-macro-report

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

heat-exchanger-model

Define heat-exchanger core model.

heat-exchanger-report

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

heat-exchanger-zone

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

plot-NTU

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

write-NTU

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

multiphase/

Enter the multiphase model menu.

body-force-formulation

Specify body force formulation.

coupled-level-set

Enable coupled level set interface tracking method.

boiling-model-options

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

eulerian-parameters

Specify Eulerian parameters.

mixture-parameters

Specify mixture parameters.

model

Specify multiphase model.

number-of-phases

Specify the number of phases.

options

Volume fraction parameters.

volume-fraction-parameters

Specify volume fraction parameters.

wet-steam/

Enter the wet steam model menu.

compile-user-defined-wetsteam-functions

Compile user-defined wet steam library.

enable?

Enable/disable the wet steam model.

load-unload-user-defined-wetsteam-library

Load or unload user-defined wet steam library.

set/

Enter the set menu for setting wet steam model options.

max-liquid-mass-fraction

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

noniterative-time-advance?

Enable/disable noniterative time advancement scheme.

nox?

Enable/disable the NOx model.

nox-parameters/

Enter the NOx parameters menu.

inlet-diffusion?

Enable/disable inclusion of diffusion at inlets.

nox-chemistry

Select NOx chemistry model.

nox-expert

Select additional NOx equations.

nox-turbulence-interaction

Set NOx turbulence interaction model.

radiation/

Enter the radiation models menu.

blending-factor

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

discrete-ordinates?

Enable/disable discrete ordinates radiation model.

discrete-transfer?

Enable/disable discrete transfer radiation model.

do-coupling?

Enable/disable DO/energy coupling.

do-irradiation?

Enable/disable the DO irradiation model.

dtrm-parameters/

Enter the dtrm parameters menu.

check-ray-file

Read DTRM rays file.

controls

Set dtrm solution controls.

make-globs

Make globs (coarser mesh) for radiation.

ray-trace

Create DTRM rays for radiation.

fast-second-order-discrete-ordinate?

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

method-partially-specular-wall

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

non-gray-model-parameters

Set parameters for non-gray model.

p1?

Enable/disable P1 radiation model.

radiation-iteration-parameters

Set iteration parameters for radiation models.

radiation-model-parameters

Set parameters for radiation models.

rosseland?

Enable/disable Rosseland radiation model.

s2s?

Enable/disable S2S radiation model.

s2s-parameters/

Enter the S2S parameters menu.

compute-fpsc-values

Compute only fpSC values based on current settings

compute-vf-only

Compute/write view factors only.

compute-write-vf

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

non-participating-boundary-zones-temperature

Set temperature for the non-participating boundary zones.

print-thread-clusters

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

print-zonewise-radiation

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

read-vf-file

Read S2S file.

set-global-faces-per-surface-cluster

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

set-vf-parameters

Set the parameters needed for the viewfactor calculations.

split-angle

Set split angle for the clustering algorithm.

use-new-cluster-algorithm

Use the new surface clustering algorithm.

use-old-cluster-algorithm

Use the old surface clustering algorithm.

solar?

Enable/disable solar model.

solar-calculator

Calculate sun direction and intensity.

solar-parameters/

Enter the solar parameters menu.

autoread-solar-data

Set autoread solar data parameters.

autosave-solar-data

Set autosave solar data parameters.

ground-reflectivity

Set ground reflectivity parameters.

illumination-parameters

Set illumination parameters.

iteration-parameters

Set update parameters.

quad-tree-parameters

Set quad-tree refinement parameters.

scattering-fraction

Set scattering fraction parameters.

sol-adjacent-fluidcells

Set solar load on for adjacent fluid cells.

sol-camera-pos

Set camera position based on sun direction vector.

sol-on-demand

Set solar load on demand.

sun-direction-vector

Set sun direction vector.

use-direction-from-sol-calc

Set direction computed from solar calculator.

solution-method-for-do-coupling

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

wsggm-cell-based

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

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.

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-cacl-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 - ε turbulence model.

ke-realizable?

Enable/disable the realizable k - ε turbulence model.

ke-rng?

Enable/disable the RNG k - ε turbulence model.

ke-standard?

Enable/disable the standard k - ε turbulence model.

k-kl-w?

Enable/disable the k-kl- ω turbulence model.

kw-easm?

Enable/disable the EASM k - ω turbulence model.

kw-low-re-correction?

Enable/disable the k - ω low Re option.

kw-shear-correction?

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

kw-sst?

Enable/disable the SST k - ω turbulence model.

kw-standard?

Enable/disable the standard k - ω turbulence model.

laminar?

Enable/disable laminar flow model.

large-eddy-simulation?

Enable/disable large eddy simulation.

les-subgrid-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 $S-\Omega$ 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 $k-\varepsilon$ 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 - ε turbulence model.

low-re-ke-index

Specify which low-Reynolds-number k - ε model is to be used. Six models are available:

Index	Model
0	Abid
1	Lam-Bremhorst
2	Launder-Sharma
3	Yang-Shih
4	Abe-Kondoh-Nagano
5	Chang-Hsieh-Chen

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

non-newtonian-modification?

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

restore-sst-v61?

Enable/disable SST formulation of v6.1.

rke-cmu-rotation-term?

Modify the C_μ definition for the realizable k - ε model.

Important

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

thermal-p-function?

Enable/disable Jayatilleke P function.

turb-non-newtonian?

Enable/disable turbulence for non-Newtonian fluids.

turbulence-damping?

Enable/disable turbulence damping and set turbulence damping parameters.

turb-pk-compressible?

Enable/disable turbulent production due to compressible divergence.

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 ω -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.

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/

Change the current graphics driver.

gl

IRIS GL

null

No graphics driver.

opengl

OpenGL

pex

X11 PEX

hbx

HP Starbase

x11

X11

xgl

Sun XGL

msw

Microsoft Windows

face-displacement

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

hidden-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-dislay-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.

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:exit / close-fluent

exit

Exit program.

close-fluent

(ANSYS Fluent in Workbench only) Exit program.

Chapter 5: file/

async-optimize?

Choose whether to optimize file I/O using scratch disks and asynchronous operations.

auto-save/

Enter the auto save menu.

append-file-name-with

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

case-frequency

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

data-frequency

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

max-files

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

overwrite-existing-files

Overwrite existing files when files are automatically saved.

retain-most-recent-files

Set autosave to retain the 5 most recent files.

root-name

Specify the root name for the files that are saved.

binary-files?

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

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.

nastran

Write a NASTRAN file.

particle-history-data

Export particle-history data.

patran-neutral

Write a PATRAN neutral file.

patran-nodal

Write a PATRAN nodal results file.

radtherm

Export RADTHERM file.

tecplot

Write a Tecplot+3DV format file.

fsi/

Enter the fluid-structure interaction menu.

display-fsi-mesh

Display the mesh for a fluid-structure interaction.

read-fsi-mesh

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

write-fsi-mesh

Write a fluid-structure interaction mesh file.

import/

Import case and data information.

abaqus/

Import an ABAQUS file.

fil

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

input

Read an ABAQUS input file as a case file.

odb

Read an ABAQUS odb file as a case file.

cfx/

Import a CFX file.

definition

Read a CFX definition file as a case file.

result

Read a CFX definition file as a case file.

cgns/
Import a CGNS file.

data
Read data from CGNS file.

mesh
Import a CGNS mesh file.

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

chemkin-mechanism
Read a CHEMKIN mechanism file.

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

ensight
Read an EnSight file as a case file.

fidap
Import a FIDAP neutral file.

flamelet/
Import a flamelet file.

standard
Read a standard format flamelet file.

oppdif
Read an OPPDIF format flamelet file.

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

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

gambit
Import a GAMBIT neutral file.

hypermesh
Read a HYPERMESH file as a case file.

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.

solution-files/

Enter the solution files menu.

delete-solution

Delete solution files.

load-solution

Load a solution file.

print-solution-files

Print a list of available solution files.

start-journal

Start recording all input in a file.

start-transcript

Start recording input and output in a file.

stop-journal

Stop recording input and close journal file.

stop-macro

Stop recording input to a macro.

stop-transcript

Stop recording input and output and close transcript file.

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.

cfld-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.

cfld-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:parallel/

bandwidth

Show network bandwidth.

gpgpu/

Enter the GPGPU menu

select

Select which GPGPUs to use for AMG acceleration

show

list the available GPGPUs. GPGPUs selected for use are indicated by the presence of an asterisk (*).

latency

Show network latency.

load-balance

Enter the load balancing parameters menu.

physical-models

Use physical-models load balancing?

dynamic-mesh

Use load balancing for dynamic mesh?

mesh-adaption

Use load balancing for mesh adaption?

network/

Enter the network configuration menu.

kill-all-nodes

Delete all compute nodes from virtual machine.

kill-node

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

load-hosts

Input a hosts database file.

path

Specify the path to the `v150/fluvent` installation directory. For most cases, the path should never have to be set.

save-hosts

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

spawn-node

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

partition/

Enter the partition domain menu.

auto/

Set auto partition parameters.

across-zones

Enable auto partitioning by zone or by domain.

load-vector

Set the auto partition load vector.

method

Set the partition method.

pre-test

Set auto partition pre-testing optimization.

use-case-file-method

Use partitions in a 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 2π .

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.

convergence/

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].

convergence-acceleration-for-stretched-meshes/

Enable convergence acceleration for stretched meshes to improve the convergence of the implicit density based solver on meshes with high cell stretching.

correction-tolerance/

Enter the correction tolerance menu.

coupled-vof-expert

Set coupled vof expert controls. You will be prompted with the following questions:

Use linearized buoyancy force?

provides the implicit linearization of buoyancy force.

Use blended treatment for buoyancy force?

will turn off buoyancy linearization in certain unstable conditions.

Use false time step linearization?

provides additional stability for buoyancy driven flows in the steady state pseudo-transient mode by increasing the diagonal dominance using false time step size.

Use smoothed density for pseudo-transient method?

smooths the cell density near the interface, 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:

Index	Model
10	Standard
11	Linear
12	Second Order
13	Body Force Weighted
14	PRESTO!

mp

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

Index	Model
0	First Order
1	Second Order
28	Compressive
5	Modified HRIC
29	BGM
4	QUICK

mom

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

Index	Model
0	First Order Upwind
1	Second Order Upwind
2	Power Law
4	QUICK
5	Third-Order MUSCL

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

Contact your ANSYS Fluent technical support engineer for more details.

equations/

Select the equations to be solved.

expert

Set expert options.

extrapolate-eqn-vars/

Enter the extrapolation menu.

extrapolate-vars?

Applies a predictor algorithm for computing initial conditions at time step n+1.

flow-warnings?

Specify whether or not to print warning messages when reversed flow occurs at inlets and outlets, and when mass flow inlets develop supersonic regions. By default, flow warnings are printed.

flux-type

Set the flux type.

gradient-scheme

Set gradient options.

heterogeneous-stiff-chemistry

Set the heterogeneous stiff-chemistry solver.

high-order-term-relaxation/

Enter the High Order Term Relaxation menu.

enable?

Enable/Disable High Order Term Relaxation.

options/

High Order Term Relaxation Options.

relaxation-factor

Set the relaxation factor.

variables/

Select the variables.

limiter-warnings?

Specify whether or not to print warning messages when quantities are being limited. By default, limiter warnings are printed.

limits

Set solver limits for various solution variables, in order to improve the stability of the solution.

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:

Index	VOF Equation Discretization	Option
0	mass conservative	reference density for a particular phase in a cell is treated as the volume averaged density of that phase in the whole domain
1	mass conservative	reference density for a particular phase in a cell is treated as the density of that phase in that cell
2	mass conservative	reference density for any phase in a cell is treated as the mixture density of that phase in that cell
3	volume conservative	reference density for a particular phase in a cell is treated as the density of that phase in that cell

max-corrections/

Enter the set max-corrections menu.

max-flow-time

Set the maximum flow time.

max-iterations-per-time-step

Set the number of time steps for a transient simulation.

Note

This option is available when automatic initialization and case modification is enabled.

multi-grid-amg

Set the parameters that govern the algebraic multigrid procedure.

multi-grid-controls/

Set multigrid parameters and termination criteria.

multi-grid-fas

Set the parameters that control the FAS multigrid solver. This command appears only when the explicit coupled solver is used.

multi-stage

Set the multi-stage coefficients and the dissipation and viscous evaluation stages. This command appears only when the explicit coupled solver is used.

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:

Index	Damping Function
0	Linear
1	Quadratic
2	Cubic
3	Cosine

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:

Index	Model
20	SIMPLE
21	SIMPLEC
22	PISO
24	Coupled

phase-based-vof-discretization

Set phase based slope limiter for VOF compressive scheme.

poor-mesh-numerics/

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 Δt .

unsteady-statistics-cff

Unsteady statistics for custom field functions.

under-relaxation/

Enter the under-relaxation menu, which allows you to set the under-relaxation factor for each equation that is being solved in a segregated manner.

undo-timestep

When enabled, if the truncation error within a time step exceeds the specified tolerance Fluent will automatically undo the current calculation and make another attempt with the time step reduced by 1/2. This will be attempted up to 5 times after which Fluent will accept the result and proceed to the next time step.

variable-time-stepping

Set variable time-stepping options for VOF explicit schemes.

vof-numerics

Set VOF numeric options.

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.

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](#).

