Ansys
2021/R2
Engineering What's Ahead.

Ansys Fluent Text Command List

Ansys

Table of Contents

List of Figures

Part I: Introduction

The section describes the general aspects of the text command interface for Fluent.

- Using This Manual (p. 3) gives a brief description of various chapters in this manual.

- Text User Interface (p. 7) gives an overview of the text command language, including basic syntax and capabilities.

- Appendix A: Text Command List Changes in Ansys Fluent 2021 R2 (p. 21) lists changes to the Text Command List for the current release.

Chapter 1: Using This Manual

> **Important:**
>
> Under U.S. and international copyright law, Ansys, Inc. is unable to distribute copies of the papers listed in the bibliography, other than those published internally by Ansys, Inc. Use your library or a document delivery service to obtain copies of copyrighted papers.

This manual describes the text-based user interface of Ansys Fluent Meshing which may be used for scripting and other advanced workflows. Here is what you will find in each chapter and appendix:

Introduction:

- Text User Interface (p. 7) gives an overview of the text command language, including basic syntax and capabilities.

- Appendix A: Text Command List Changes in Ansys Fluent 2021 R2 (p. 21) lists changes to the Text Command List for the current release.

Meshing Mode:

- `boundary/` (p. 25) lists the commands used for creating, managing, modifying, deleting boundary (face) zones and surface mesh.

- `cad-assemblies/` (p. 55) lists commands used to interact with your CAD model.

- `diagnostics/` (p. 59) lists commands used to find and repair face zone problems related to connectivity and quality.

- `display/` (p. 63) lists commands used to control the rendering of your model and mesh in the graphical window.

- `exit` (p. 87) lists the command to close the program.

- `file/` (p. 89) lists commands used to input and output your data.

- `material-point/` (p. 103) lists commands used to manage material points in your model.

- `mesh/` (p. 105) lists commands used to create, manage, modify and delete cell zones and volume mesh.

- `objects/` (p. 151) lists commands used in the object-based meshing workflow, from CAD import to volume fill.

- `parallel/` (p. 163) lists commands specific to parallel processing.

- `preferences/` (p. 165) lists commands specific to user preferences.

- `report/` (p. 177) lists commands used to return volume and surface mesh statistics such as counts or quality.

- `scoped-sizing/` (p. 181) lists commands used to create and manage scoped size controls.

- `size-functions/` (p. 183) lists commands used to create and manage size functions such as global or periodic sizes.

- `switch-to-solution-mode` (p. 187) lists the command to transfer your mesh data to the Fluent solver.

- Appendix A: Query and Utility Functions (p. 189) lists API functions used to interact with your model by passing information such as names or lists of zones or objects, statistics, and diagnostics. Boolean operations and examples are included.

- Appendix B: Boundary Functions (p. 219) lists API functions used to manage boundary (face) zone mesh. Examples are included.

- Appendix C: Connect Functions (p. 233) lists API functions used to manage boundary (face) zone connectivity. Examples are included.

- Appendix D: Size Field Functions (p. 237) lists API functions used to manage geodesic size controls and size fields. Examples are included.

- Appendix E: Wrap Functions (p. 247) lists API functions used to manage your mesh in UTM cases. Examples are included.

Solution Mode:

- `adjoint/` (p. 259) lists commands related to the adjoint solver.

- `define/` (p. 271) lists commands related to problem definition, such as models, boundary conditions, materials, etc.

- `display/` (p. 383) lists commands used to control the rendering of your model and mesh in the graphical window.

- `exit / close-fluent` (p. 411) lists the command to close the program.

- `file/` (p. 413) lists commands used to input and output your data.

- `mesh/` (p. 431) lists commands used to create and manage mesh properties.

- `parallel/` (p. 441) lists commands specific to parallel processing.

- `plot/` (p. 447) lists commands specific to plotting data.

- `preferences/` (p. 453) lists commands for controlling preferences.

- `report/` (p. 465) lists commands used to return statistics for the simulation.

- `server/` (p. 475) lists commands used control the Fluent Remote Visualization Client and Server.

- `solve/` (p. 477) lists commands used to create and manage solution controls, such as animation, cell registers, monitors, initialization, etc.

- `surface/` (p. 517) lists commands related to creating and manipulating surfaces.

- `switch-to-meshing-mode` (p. 521) lists the command to transfer your solution data to Fluent in meshing mode.

- `turbo-post/` (p. 523) lists commands related to results and reporting for turbomachinery.

- `views/` (p. 525) lists commands related to camera and view manipulation in the graphics window.

- Battery Model Text Commands (p. 527) lists commands related to battery models.

- Fuel Cell Text Commands (p. 529) lists commands related to fuel cell models.

- Magnetohydrodynamics Text Commands (p. 541) lists commands related to magnetohydrodynamics models.

- `continuous-fiber/` (p. 543) lists commands related to the continuous fiber model.

Chapter 2: Text User Interface

In addition to the graphical user interface, the meshing mode of Ansys Fluent also consists of a textual command line reference.

The text user interface (TUI) is written in a dialect of Lisp called Scheme. Users familiar with Scheme will be able to use the interpretive capabilities of the interface to create customized commands. The TUI is described in the following sections:

2.1. Text Menu System

The text menu system provides a hierarchical interface to the program's underlying procedural interface.

- You can easily manipulate its operation with standard text-based tools: input can be saved in files, modified with text editors, and read back in to be executed, because it is text based.

- The text menu system is tightly integrated with the Scheme extension language, so it can easily be programmed to provide sophisticated control and customized functionality.

> **Note:**
>
> The console also contains an automatic-completer feature that shows the valid inputs as soon as you begin typing so that you can quickly enter your intended commands. This feature can be turned on/off in **Preferences**.

The menu system structure is similar to the directory tree structure of LINUX operating systems. When you first start Fluent, you are in the "root" menu and the menu prompt is simply a caret/greater-than symbol:

```
>
```

To generate a listing of the submenus and commands in the current menu, press **Enter**. The available submenus and commands will depend on whether you are running Fluent in meshing mode or solution mode.

For instance, the submenus and commands that are available from the root menu of the solution mode are as follows:

```
>Enter
adapt/                  mesh/                   surface/
define/                 parallel/               switch-to-meshing-mode
display/                plot/                   views/
exit                    report/
file/                   solve/
```

By convention, submenu names end with a `/` to differentiate them from menu commands. To execute a command, type its name (or an abbreviation). Similarly, to move down into a submenu, enter its name or an abbreviation. When you move into the submenu, the prompt will change to reflect the current menu name.

```
> display

/display > set

/display/set >
```

To move back to the previously occupied menu, type `q` or `quit` at the prompt.

```
/display/set > q

/display
```

You can move directly to a menu by giving its full pathname.

```
/display > /file

/display//file >
```

In the above example, control was passed from `/display` to `/file` without stopping in the root menu. Therefore, when you quit from the `/file` menu, control will be passed directly back to `/display`.

```
/display//file > q

/display >
```

If you execute a command without stopping in any of the menus along the way, control will again be returned to the menu from which you invoked the command.

```
/display /file start-journal jrnl

/display >
```

The text menu system provides online help for menu commands. The text menu online help system is described in Using the Text Interface Help System (p. 18).

To edit the current command, you can position the cursor with the left and right arrow keys, delete with the **Backspace** key, and insert text simply by typing.

For additional information, refer to the following sections:

2.1.1. Command Abbreviation

2.1.2. Command Line History

2.1.3. Scheme Evaluation

2.1.4. Aliases

2.1.1. Command Abbreviation

To select a menu command, you need not type the entire name; you can type an abbreviation that matches the command.

- A command name consists of "phrases" separated by hyphens.

- A command is matched by matching an initial sequence of its phrases.

- Matching of hyphens is optional.

- A phrase is matched by matching an initial sequence of its characters.

- A character is matched by typing that character.

The rules for "matching" a command are:

- If an abbreviation matches more than one command, then the command with the greatest number of matched phrases is chosen.

- If more than one command has the same number of matched phrases, then the first command to appear in the menu is chosen.

  For example, each of the following will match the given command `set-ambientcolor`: `set-ambient-color`, `s-a-c`, `sac`, and `sa`.

- When abbreviating commands, sometimes your abbreviation will match more than one command. In such cases, the first command is selected.

- Occasionally, there is an anomaly such as `lint` not matching `lighting-interpolation` because the `li` gets absorbed in `lights-on?` and then the `nt` does not match `interpolation`.

  This can be resolved by choosing a different abbreviation, such as `liin`, or `l-int`.

2.1.2. Command Line History

You can use the up and down arrow keys on your keyboard to go through recently used commands that are stored in history. By default, command-history will store only the last ten commands. This can be changed (for example to 15) by using the following command:

```
> (set! *cmd-history-length* 15)
```

> **Important:**
>
> Command-history is not available if the Ansys Fluent application is started with -g options (see Command Line Startup Options in the *Fluent User's Guide*).

> **Important:**
>
> The user inputs supplied as the arguments of the TUI command or alias will not be saved in history. By way of illustration, consider the following entry in the TUI:

```
> rc new_file.cas
```

Important:

In history, only `rc` (an alias for `read-case`) will be saved, since `new_file.cas` is a user input to the alias-function.

Commands recalled from history can be edited or corrected using the **Backspace** key and the left and right arrow keys.

2.1.3. Scheme Evaluation

If you enter an open parenthesis, `(`, at the menu prompt, then that parenthesis and all characters up to and including the matching closing parenthesis are passed to Scheme to be evaluated, and the result of evaluating the expression is displayed.

```
> (define a 1)

a

> (+ a 2 3 4)

10
```

2.1.4. Aliases

Command aliases can be defined within the menu system. As with the Linux `csh` shell, aliases take precedence over command execution. The following aliases are predefined in Cortex: `error`, `pwd`, `chdir`, `ls`, `.`, and `alias`.

error

Displays the Scheme object that was the "irritant" in the most recent Scheme error interrupt.

pwd

Prints the working directory in which all file operations will take place.

chdir

Will change the working directory.

ls

Lists the files in the working directory.

. (period)

Prompts you for a journal file name and then run the journal file you have specified.

alias

Displays the list of symbols currently aliased.

2.2. Text Prompt System

Commands require various arguments, including numbers, filenames, yes/no responses, character strings, and lists. A uniform interface to this input is provided by the text prompt system. A prompt consists of a prompt string, followed by an optional units string enclosed in parentheses, followed by a default value enclosed in square brackets. The following shows some examples of prompts:

```
filled-mesh? [no] Enter

shrink-factor [0.1] Enter

line-weight [1] Enter

title [""]Enter
```

The default value for a prompt is accepted by pressing **Enter** on the keyboard or typing a `,` (comma).

Important:

Note that a comma is not a separator. It is a separate token that indicates a default value. The sequence "`1,2`" results in three values; the number 1 for the first prompt, the default value for the second prompt, and the number 2 for the third prompt.

A short help message can be displayed at any prompt by entering a `?`. (See Using the Text Interface Help System (p. 18).)

To abort a prompt sequence, simply press **Ctrl**+**c**.

2.2.1. Numbers

The most common prompt type is a number. Numbers can be either integers or real numbers. Valid numbers are, for example, `16`, `-2.4`, `.9e5`, and `+1e-5`.

- Integers can also be specified in binary, octal, and hexadecimal form.

- The decimal integer 31 can be entered as `31`, `#b11111`, `#o37`, or `#x1f`.

- In Scheme, integers are a subset of reals, so you do not need a decimal point to indicate that a number is real; `2` is just as much a real as `2.0`.

- If you enter a real number at an integer prompt, any fractional part will be truncated. For example, `1.9` will become `1`.

2.2.2. Booleans

Some prompts require a yes-or-no response. A yes/no prompt will accept either `yes` or `y` for a positive response, and `no` or `n` for a negative response. Yes/no prompts are used for confirming potentially

dangerous actions such as overwriting an existing file, exiting without saving case, data, mesh, and so on.

Some prompts require actual Scheme Boolean values (true or false). These are entered with the Scheme symbols for true and false, `#t` and `#f`.

2.2.3. Strings

Character strings are entered in double quotes, for example, `"red"`. Plot titles and plot legend titles are examples of character strings. Character strings can include any characters, including blank spaces and punctuation.

2.2.4. Symbols

Symbols are entered *without* quotes. Zone names, surface names, and material names are examples of symbols. Symbols must start with an alphabetical character (that is, a letter), and cannot include any blank spaces or commas.

You can use wild cards to specify zone names when using the TUI. Some examples are:

- `*` will translate as "all zones".

  For example,

  – `/display/boundary-grid *` enables you to display all the boundary zones in the mesh.

  – `/boundary/delete-island-faces wrap*` enables you to delete island faces on all zones prefixed by `wrap`.

- `>` will translate as "all zones visible in the graphics window".

  For example, `/boundary/manage/delete >, yes` enables you to delete all visible zones.

- `^` will translate as "all zones selected in the graphics window".

  For example, `/boundary/manage/delete ^, yes` enables you to delete all selected zones.

- `[object_name` will translate as "all zones with the name *object_name*".

  For example, `/boundary/manage/delete [box, yes` enables you to delete all zones of an object with the name *box*.

- `[object_name/label_name` will translate as "all zones with *label_name* of *object_name*"

  For example, `/boundary/manage/delete [fluid/box*, yes` enables you to delete all zones of an object with the name *fluid* and comprising face zone labels *box**.

If you use a wild card for an operation that requires a single zone as input, you will be prompted to specify a single zone from the list of those that match the expression specified.

```
> /boundary/manage/name wall*   <Enter>

wall-1 wall-3 wall-5
```

```
wall-2 wall-4 wall-6
Zone Name [ ]
```

2.2.5. Filenames

Filenames are actually just character strings. For convenience, filename prompts do not require the string to be surrounded with double quotes. If, for some exceptional reason, a filename contains an embedded space character, then the name must be surrounded with double quotes.

One consequence of this convenience is that filename prompts do not evaluate the response. For example, the sequence

```
> (define fn "valve.ps")

fn

> hc fn
```

Will end up writing a picture file with the name `fn`, not `valve.ps`. Since the filename prompt did not evaluate the response, `fn` did not get a chance to evaluate "`valve.ps`" as it would for most other prompts.

2.2.6. Lists

Some functions in Ansys Fluent require a "list" of objects such as numbers, strings, Booleans, and so on. A list is a Scheme object that is simply a sequence of objects terminated by the empty list, `'()`. Lists are prompted for an element at a time, and the end of the list is signaled by entering an empty list. This terminating list forms the tail of the prompted list, and can either be empty or can contain values. For convenience, the empty list can be entered as `()` as well as the standard form `'()`. Normally, list prompts save the previous argument list as the default. To modify the list, overwrite the desired elements and terminate the process with an empty list. For example,

```
element(1) [()] 1

element(2) [()] 10

element(3) [()] 100

element(4) [()] Enter
```

Creates a list of three numbers: 1, 10, and 100. Subsequently,

```
element(1) [1] Enter

element(2) [10] Enter

element(3) [100] Enter

element(4) [()] 1000

element(5) [()] Enter
```

Adds a fourth element. Then

```
element(1) [1] Enter

element(2) [10] Enter
```

```
element(3) [100] ()
```

Leaves only 1 and 10 in the list. Subsequently entering

```
element(1) [1] ,,'(11 12 13)
```

Creates a five element list: 1, 10, 11, 12, and 13. Finally, a single empty list removes all elements

```
element(1) [1] ()
```

A different type of list, namely, a "list-of-scalars" contains pick menu items (and not list items) for which a selection has to be made from listed quantities, which are available at the **Enter** prompt. Hence, a list-of-scalars cannot be entered as a list.

An example of a "list-of-scalars" consists of the following:

```
ASCII scalar(1)>
abs-angular-coordinate   cell-warp              radial-velocity
absolute-pressure        cell-weight            rel-tangential-velocity
adaption-curvature       cell-zone              rel-total-pressure
adaption-function        custom-function-0      rel-velocity-magnitude
adaption-iso-value       density                relative-velocity-angle
adaption-space-gradient  density-all            relative-x-velocity
angular-coordinate       dp-dx                  relative-y-velocity
axial-coordinate         dp-dy                  relative-z-velocity
axial-velocity           dp-dz                  tangential-velocity
boundary-cell-dist       dynamic-pressure       total-pressure
boundary-normal-dist     existing-value         velocity-angle
boundary-volume-dist     face-area-magnitude    velocity-magnitude
cell-children            face-handedness        x-coordinate
cell-element-type        face-squish-index      x-face-area
cell-equiangle-skew      interface-wall-zone    x-velocity
cell-equivolume-skew     mass-imbalance         x-velocity-residual
cell-partition           mesh-x-velocity        y-coordinate
cell-refine-level        mesh-y-velocity        y-face-area
cell-squish-index        mesh-z-velocity        y-velocity
cell-surface-area        partition-neighbors    y-velocity-residual
cell-type                pressure               z-coordinate
cell-volume              pressure-coefficient   z-face-area
cell-volume-change       pressure-residual      z-velocity
cell-wall-distance       radial-coordinate      z-velocity-residual

ASCII scalar(1)> pressure

ASCII scalar(2)> z-coordinate

ASCII scalar(3)> cell-volume

ASCII scalar(4)> dynamic-pressure velocity-magnitude x-face-area

ASCII scalar(7)> █
```

2.2.7. Evaluation

All responses to prompts (except filenames, see above) are evaluated by the Scheme interpreter before they are used. You can therefore enter any valid Scheme expression as the response to a prompt. For example, to enter a unit vector with one component equal to 1/3 (without using your calculator),

```
/foo> set-xy
x-component [1.0] (/ 1 3)

y-component [0.0] (sqrt (/ 8 9))
```

Or, you could first define a utility function to compute the second component of a unit vector,

```
> (define (unit-y x) (sqrt (- 1.0 (* x x))))

unit-y
```

```
/foo> set-xy

x-component [1.0] (/ 1 3)

y-component [0.0] (unit-y (/ 1 3))
```

2.2.8. Default Value Binding

The default value at any prompt is bound to the Scheme symbol " _" (underscore) so that the default value can form part of a Scheme expression. For example, if you want to decrease a default value so that it is one-third of the original value, you could enter

```
shrink-factor [0.8] (/ _ 3)
```

2.3. Interrupts

The execution of the code can be halted by pressing the **Ctrl**+**c**, at which time the present operation stops at the next recoverable location.

2.4. System Commands

The way you execute system commands with the ! (bang) shell escape character will be slightly different for Linux and Windows systems.

For additional information, see the following sections:

2.4.1. System Commands for LINUX-based Operating Systems
2.4.2. System Commands for Windows-based Operating Systems

2.4.1. System Commands for LINUX-based Operating Systems

If you are running Ansys Fluent under a Linux-based operating system, all characters following the ! up to the next newline character will be executed in a subshell. Any further input related to these system commands must be entered in the window in which you started the program, and any screen output will also appear in that window. (Note that if you started Ansys Fluent remotely, this input and output will be in the window in which you started Cortex.)

```
> !rm junk.*

> !vi script.rp
```

!pwd and !ls will execute the Linux commands in the directory in which Cortex was started. The screen output will appear in the window in which you started Ansys Fluent, unless you started it remotely, in which case the output will appear in the window in which you started Cortex. (Note that !cd executes in a subshell, so it will not change the working directory either for Ansys Fluent or for Cortex, and is therefore not useful.) Typing cd with no arguments will move you to your home directory in the console.

Ansys Fluent includes three system command aliases (pwd, ls, and chdir) that will be executed in your working directory with output displayed in the Ansys Fluent console. Note that these aliases will invoke the corresponding Linux commands with respect to the parent directory of the case file. For

example, `pwd` prints the parent directory of the case file in the Ansys Fluent console, while `!pwd` prints the directory from which you started Ansys Fluent in the Linux shell window where you started Ansys Fluent.

Several examples of system commands entered in the console are shown below. The screen output that will appear in the window in which Ansys Fluent was started (or, if you started the program remotely, in the window in which Cortex was started) follows the examples.

Example input (in the Ansys Fluent console):

```
> !pwd

> !ls valve*.*
```

Example output (in the window in which Ansys Fluent— or Cortex, if you started the program remotely—was started):

```
/home/cfd/run/valve
 valve1.cas   valve1.msh   valve2.cas   valve2.msh
```

2.4.2. System Commands for Windows-based Operating Systems

If you are running Ansys Fluent under a Windows operating system, all characters following the `!` up to the next newline character will be executed. The results of a command will appear in the Ansys Fluent console, or in a separate window if the command starts an external program, such as Notepad.

```
> !del junk.*

> !notepad script.rp
```

`!cd` and `!dir` will execute the DOS commands and the screen output will appear in the Ansys Fluent console. The `!cd` command with no argument will display the current working directory in the Ansys Fluent console.

Several examples of system commands entered in the console are shown below.

Example input (in boxes) and output (in the Ansys Fluent console):

```
 > !cd

p:/cfd/run/valve
 > !dir valve*.*/w

    Volume in drive P is users
    Volume Serial Number is 1234-5678
    Directory of p:/cfd/run/valve
    valve1.cas       valve1.msh       valve2.cas       valve2.msh
       4 File(s)            621,183 bytes
       0 Dir(s)         1,830,088,704 bytes free
```

2.5. Text Menu Input from Character Strings

When writing a Scheme extension function for Ansys Fluent, it is often convenient to be able to include menu commands in the function. This can be done with `ti-menu-load-string`. For example, to open graphics window 2, use

```
(ti-menu-load-string "di ow 2")
```

A Scheme loop that will open windows 1 and 2 and display the front view of the mesh in window 1 and the back view in window 2 is given by

```
(for-each
 (lambda (window view)
    (ti-menu-load-string (format #f "di ow ~a gr view rv ~a"
 window view)))
 '(1 2)
 '(front back))
```

This loop makes use of the `format` function to construct the string used by `menu-load-string`. This simple loop could also be written without using menu commands at all, but you need to know the Scheme functions that get executed by the menu commands to do it:

```
(for-each
 (lambda (window view)
    (cx-open-window window)
    (display-mesh)
    (cx-restore-view view))
 '(1 2) '(front back))
```

String input can also provide an easy way to create aliases within Ansys Fluent. For example, to create an alias that will display the mesh, you could type the following:

```
(alias 'dg (lambda () (ti-menu-load-string "/di gr")))
```

Then any time you enter `dg` from anywhere in the menu hierarchy, the mesh will be drawn in the active window.

Important:

`ti-menu-load-string` evaluates the string argument in the top level menu. It ignores any menu you may be in when you invoke `ti-menu-load-string`.

As a result, the command

```
(ti-menu-load-string "open-window 2 gr") ; incorrect usage
```

Will not work even if you type it from within the `display/` menu—the string itself must cause control to enter the `display/` menu, as in

```
(ti-menu-load-string "display open-window 2 mesh")
```

2.6. Using the Text Interface Help System

The text user interface provides context-sensitive online help. Within the text menu system, you can obtain a brief description of each of the commands by entering a `?` followed by the command in question.

Example:

```
 > ?dis
display/: Enter the display menu.
```

You can also enter a lone `?` to enter "help mode." In this mode, you need only enter the command or menu name to display the help message. To exit the help mode type `q` or `quit` as for a normal menu.

Example:

```
 > ?

[help-mode]> di

display/: Enter the display menu.

 [help-mode]> pwd

 pwd: #[alias]
 (LAMBDA ()
   (BEGIN
   (SET! pwd-cmd ((LAMBDA n
 n) 'system (IF (cx-send '(unix?))
     "pwd"
     "cd")))
  (cx-send pwd-cmd)))

 [help-mode]> q
```

To access the help, type a `?` at the prompt when you are prompted for information.

Example:

```
> display/annotate Annotation text [""] ?
 Annotation text [""]
```

Appendix A. Text Command List Changes in Ansys Fluent 2021 R2

For a complete listing of changes to the Text Command List for Ansys Fluent 2021 R2, refer to Text Command Changes in Ansys Fluent 2021 R2 in the Fluent Migration Manual.

Part II: Meshing Mode

The section describes the text command listing for Fluent in Meshing mode.

- `boundary/` (p. 25) lists the commands used for creating, managing, modifying, deleting boundary (face) zones and surface mesh.

- `cad-assemblies/` (p. 55) lists commands used to interact with your CAD model.

- `diagnostics/` (p. 59) lists commands used to find and repair face zone problems related to connectivity and quality.

- `display/` (p. 63) lists commands used to control the rendering of your model and mesh in the graphical window.

- `exit` (p. 87) lists the command to close the program.

- `file/` (p. 89) lists commands used to input and output your data.

- `material-point/` (p. 103) lists commands used to manage material points in your model.

- `mesh/` (p. 105) lists commands used to create, manage, modify and delete cell zones and volume mesh.

- `objects/` (p. 151) lists commands used in the object-based meshing workflow, from CAD import to volume fill.

- `parallel/` (p. 163) lists commands specific to parallel processing.

- `preferences/` (p. 165) lists commands specific to user preferences.

- `report/` (p. 177) lists commands used to return volume and surface mesh statistics such as counts or quality.

- `scoped-sizing/` (p. 181) lists commands used to create and manage scoped size controls.

- `size-functions/` (p. 183) lists commands used to create and manage size functions such as global or periodic sizes.

- `switch-to-solution-mode` (p. 187) lists the command to transfer your mesh data to the Fluent solver.

- Appendix A: Query and Utility Functions (p. 189) lists API functions used to interact with your model by passing information such as names or lists of zones or objects, statistics, and diagnostics. Boolean operations and examples are included.

- Appendix B: Boundary Functions (p. 219) lists API functions used to manage boundary (face) zone mesh. Examples are included.

- lists API functions used to manage boundary (face) zone connectivity. Examples are included.

- lists API functions used to manage geodesic size controls and size fields. Examples are included.

- lists API functions used to manage your mesh in UTM cases. Examples are included.

Chapter 1: `boundary/`

`boundary/auto-slit-faces`

Slits all boundary faces with cells on both sides (these cells must be in the same cell zone). A displacement can be specified to provide thickness to the boundary.

`boundary/boundary-conditions/`

Contains options for copying or clearing boundary conditions when a case file is read.

> **`boundary/boundary-conditions/clear`**
>
> Clears the boundary conditions assigned to the specified face zones.
>
> **`boundary/boundary-conditions/clear-all`**
>
> Clears the boundary conditions assigned to all the face zones.
>
> **`boundary/boundary-conditions/copy`**
>
> Enables you to copy the boundary conditions from the face zone selected to the face zones specified.

`boundary/check-boundary-mesh`

Reports the number of Delaunay violations on the triangular surface mesh and the number of isolated nodes.

`boundary/check-duplicate-geom`

Displays the names of the duplicate surfaces and prints maximum and average distance between them.

`boundary/clear-marked-faces`

Clears marked faces.

`boundary/clear-marked-nodes`

Clears nodes that were marked using the `mark-duplicate-nodes` command.

`boundary/compute-bounding-box`

Computes the bounding box for the zones specified.

`boundary/count-free-nodes`

Reports the number of boundary nodes associated with edges having only one attached face.

`boundary/`count-marked-faces

Reports the number of marked faces.

`boundary/`count-unused-bound-node

Counts the unused boundary nodes in the domain.

`boundary/`count-unused-faces

Lists the number of boundary faces that are not used by any cell.

`boundary/`count-unused-nodes

Lists the number of boundary nodes that are not used by any cell.

`boundary/`create-bounding-box

Creates the bounding box for the specified zones. You can specify the zone type, name, edge length, and the extents of the box, as required. You can also optionally create a geometry object from the bounding box created.

`boundary/`create-cylinder

Creates a cylinder by specifying the axis, radius, and edge length or three arc nodes, the axial delta, the radial gap, and the edge length. You can also specify the prefix for the zone being created, as required. You can also optionally create a geometry object from the cylinder created.

`boundary/`create-plane-surface

Creates a plane surface by specifying either the axis direction, axial location, and the extents of the surface or three points defining the plane. You can also optionally create a geometry object from the plane surface created.

`boundary/`create-revolved-surface

Creates a revolved surface by rotating the specified edge through the angle specified. Specify the number of segments, scale factor, and the pivot point and axis of rotation. You can also optionally create a geometry object from the revolved surface created.

`boundary/`create-swept-surface

Creates a surface by sweeping the specified edge in the direction specified. You need to specify the distance to sweep through and the number of offsets, as required. You can also optionally create a geometry object from the swept surface created.

`boundary/`delete-all-dup-faces

Searches for faces on all boundary zones that have the same nodes and deletes the duplicates.

`boundary/`delete-duplicate-faces

Searches for faces on a specified zone that have the same nodes and deletes the duplicates.

Duplicate faces may be present if you generated the boundary mesh using a third-party grid generator, or if you have used the `slit-boundary-face` command to modify the boundary mesh and then merged the nodes.

`boundary/`delete-free-edge-faces

Enables you to remove faces with the specified number of free edges from the specified boundary zones.

`boundary/`delete-island-faces

Enables you to delete faces in a non-contiguous region of a face zone.

`boundary/`delete-unconnected-faces

Enables you to delete the unconnected face-zones.

`boundary/`delete-unused-faces

Deletes all the boundary faces that are not used by any cell.

`boundary/`delete-unused-nodes

Deletes the boundary nodes that are not used by any boundary faces.

`boundary/`edge-limits

Prints the length of the shortest and longest edges on the boundary. This information is useful for setting initial mesh parameters and refinement controls.

`boundary/`face-distribution

Reports the distribution of face quality in the text window.

`boundary/`face-skewness

Lists the worst face skewness.

`boundary/`feature/

Enables you to create and modify features.

`boundary/feature/`copy-edge-zones

Copies the specified edge zone(s) to new edge zone(s).

boundary/feature/create-edge-zones

Extracts edge loops for the specified face zone(s) based on the feature method specified. You also need to specify an appropriate value for feature angle when using the `fixed-angle` method.

> **Note:**
>
> The **Face Seed** approach cannot be used when creating edge loops using text commands.

boundary/feature/delete-degenerated-edges

Deletes degenerated edges (edges where the two end nodes are the same) for the edge zone(s) specified.

boundary/feature/delete-edge-zones

Deletes the specified edge zone(s)

boundary/feature/edge-size-limits

Reports the minimum, maximum, and average edge length for the specified edge zone(s) in the console.

boundary/feature/group

Associates the specified edge zone(s) with the specified face zone.

boundary/feature/intersect-edge-zones

Intersects the specified edge loops to create a new edge loop comprising the common edges. You can enable automatic deleting of overlapped edges and specify an appropriate intersection tolerance.

boundary/feature/list-edge-zones

Lists the name, ID, type, and count for the specified edge zone(s).

boundary/feature/merge-edge-zones

Merges multiple edge loops of the same type into a single loop.

boundary/feature/orient-edge-direction

Orients the edges on the loop to point in the same direction.

boundary/feature/project-edge-zones

Projects the edges of the specified loop onto the specified face zone using the specified projection method.

boundary/feature/remesh-edge-zones

Remeshes the specified edge loop(s), modifying the node distribution according to the specified remeshing method, spacing values, and feature angle. You can also enable quadratic reconstruction, if required.

boundary/feature/reverse-edge-direction

Reverses the direction of the edge loop.

boundary/feature/separate-delete-small-edges

Separates the edge zones based on the feature angle specified, and then deletes the edges having a count smaller than the minimum count specified.

boundary/feature/separate-edge-zones

Separates the specified edge loop based on connectivity and the specified feature angle.

boundary/feature/separate-edge-zones-by-seed

Separates the edge loop based on the seed edge specified. The edge zone separation angle is used to separate the edge zone (default 40).

boundary/feature/toggle-edge-type

Toggles the edge type between boundary and interior.

boundary/feature/ungroup

Ungroups previously grouped edge zones.

boundary/fix-mconnected-edges

Resolves multi-connected edges/non-manifold configurations in the boundary mesh by deleting fringes and overlaps based on threshold values specified.

boundary/improve/

Enables you to improve boundary surfaces.

boundary/improve/collapse-bad-faces

Enables you to collapse the short edge of faces having a high aspect ratio or skewness in the specified face zone(s).

boundary/improve/degree-swap

Enables you to improve the boundary mesh by swapping edges based on a node degree value other than 6. The node degree is defined as the number of edges connected to the node.

boundary/improve/improve

Enables you to improve the boundary surface quality using skewness, size change, aspect ratio, or area as the quality measure.

boundary/improve/smooth

Enables you to improve the boundary surface using smoothing.

boundary/improve/swap

Enables you to improve the boundary surface using edge swapping.

boundary/jiggle-boundary-nodes

Randomly perturbs all boundary nodes based on an input tolerance. Some nodes will be perturbed less than the tolerance value, while others will be perturbed by half of the tolerance value in all three coordinate directions.

boundary/make-periodic

Enables you to make the specified boundaries periodic. You can specify the type of periodicity (rotational or translational), the angle, pivot, and axis of rotation, for rotational periodicity or the translational shift for translational periodicity.

For each of the zones specified, a corresponding periodic shadow boundary zone will be created.

boundary/manage/

Contains options for manipulating the boundary zones.

boundary/manage/auto-delete-nodes?

Specifies whether or not unused nodes should be deleted when their face zone is deleted.

boundary/manage/change-prefix

Enables you to change the prefix for the specified face zones.

boundary/manage/copy

Copies all nodes and faces of the specified face zone(s).

boundary/manage/create

Creates a new face zone.

boundary/manage/delete

Deletes the face zone.

boundary/manage/flip

Reverses the normal direction of the specified boundary zone(s).

boundary/manage/id

Specifies a new boundary zone ID. If there is a conflict, the change will be ignored.

boundary/manage/list

Prints information about all boundary zones.

boundary/manage/merge

Merges face zones.

boundary/manage/name

Gives a face zone a new name.

boundary/manage/orient

Consistently orients the faces in the specified zones.

boundary/manage/origin

Specifies a new origin for the mesh, to be used for face zone rotation and for periodic zone creation. The default origin is (0,0,0).

boundary/manage/remove-suffix

Removes the suffix (characters including and after the leftmost "**:**") in the face zone names.

boundary/manage/rotate

Rotates all nodes of the specified face zone(s).

boundary/manage/rotate-model

Rotates all nodes of the model through the specified angle, based on the specified point and axis of rotation.

boundary/manage/scale

Scales all nodes of the specified face zone(s).

boundary/manage/scale-model

Scales all nodes of the model by multiplying the node coordinates by the specified scale factors (x, y, z).

boundary/manage/translate

Translates all nodes of the specified face zone(s).

boundary/manage/translate-model

Translates all nodes of the model by the specified translation offsets (x, y, z).

> **Note:**
>
> The translation offsets are interpreted as absolute numbers in meshing mode. In solution mode, however, the translation offsets are assumed to be distances in the length unit set. This may lead to differences in domain extents reported after translating the mesh in the respective modes.

boundary/manage/type

Changes the boundary type of the face zone.

> **Note:**
>
> When changing the boundary type of any zone to type `interior`, ensure that there is a single cell zone across the `interior` boundary. Retaining multiple cell zones across an `interior` boundary can cause undesirable results with further tet meshing or smoothing operations.
>
> Also, face zones having no/one neighboring cell zone should not be changed to type `interior`.
>
> The mesh check will issue a warning if multiple cell zones are maintained across an `interior` boundary. The boundary type in such cases should be set to `internal` instead.

boundary/manage/user-defined-groups/

Enables you to manipulate user-defined groups.

boundary/manage/user-defined-groups/activate

Activates the specified user-defined groups.

boundary/manage/user-defined-groups/create

Creates the user-defined group comprising the specified zones.

boundary/manage/user-defined-groups/delete

Deletes the specified user-defined group.

boundary/manage/user-defined-groups/list

Lists the groups in the console.

boundary/manage/user-defined-groups/update

Enables you to modify an existing group.

boundary/mark-duplicate-nodes

Marks duplicate nodes. The marked nodes will appear in the grid display when nodes are displayed. For a list of duplicate nodes, set the `/report/verbosity` level to 2 before using the `mark-duplicate-nodes` command.

boundary/mark-face-intersection

Marks intersecting faces. Intersection is detected if the line defined by any two consecutive nodes on a face intersects any face in the current domain. The marked faces will appear in the grid display when faces are displayed. For a list of intersecting faces, set the `/report/verbosity` level to 2 before using the `mark-face-intersection` command.

boundary/mark-face-proximity

Marks faces that are in proximity to each other.

Face A is considered to be in proximity to face B if any of the nodes on face A are within the calculated proximity distance from face B. The proximity distance is calculated based on the specified relative distance and the sphere radius. The sphere radius is determined by the maximum distance from the centroid of the face to its nodes. The marked faces will appear in the grid display when faces are displayed.

For a list of faces in proximity to each other, set the `/report/verbosity` level to 2 before using the `mark-face-proximity` command.

boundary/mark-faces-in-region

Marks the faces that are contained in a specified local refinement region.

boundary/merge-nodes

Merges duplicate nodes.

boundary/merge-small-face-zones

Merges the face zones having area less than the minimum area.

boundary/modify/

Contains commands used to modify the boundary mesh.

boundary/modify/analyze-bnd-connectvty

Finds and marks free edges and nodes and multiply-connected edges and nodes. This process is necessary if the boundary mesh has been changed with Scheme functions.

boundary/modify/clear-selections

Clears all selections.

boundary/modify/clear-skew-faces

Clears faces that were marked using the `mark-skew-face` command.

`boundary/modify/`collapse

Collapses pairs of nodes, edge(s), or face(s). If a pair of nodes is selected, both the nodes are deleted and a new node is created at the midpoint of the two nodes. If a triangular face is selected, the complete face is collapsed into a single node at the centroid of the face.

`boundary/modify/`create

Creates a boundary face if the selection list contains 3 nodes and an optional zone. If the selection list contains positions, then nodes are created.

`boundary/modify/`create-mid-node

Creates a node at the midpoint between two selected nodes.

`boundary/modify/`delete

Deletes all selected faces and nodes.

`boundary/modify/`delta-move

Moves the selected node by specified magnitude.

`boundary/modify/`deselect-last

Removes the last selection from the selection list.

`boundary/modify/`hole-feature-angle

Specifies the feature angle for consideration of holes in the geometry.

`boundary/modify/`list-selections

Lists all of the selected objects.

`boundary/modify/`local-remesh

Remeshes marked faces or faces based on selections in the graphics window. Select the faces to be remeshed and specify the sizing source (`constant-size`, `geometry`, or `size-field`), the number of radial layers of faces to be remeshed (`rings`), the feature angle to be preserved while remeshing the selected faces, and the size for constant size remeshing (if applicable).

`boundary/modify/`mark-skew-face

Marks faces that should be skipped when the worst skewed face is reported using the **Modify Boundary** dialog box. This enables you to search for the next skewed face.

`boundary/modify/`merge

Merges pairs of nodes. The first node selected is retained, and the second is the duplicate that is merged.

34

boundary/modify/move

Moves the selected node to the selected position if the selection list contains a node and a position.

boundary/modify/next-skew

Finds the triangular face of nearest lower skewness value than that of the worst skewed face. The face ID, its skewness, the longest edge ID, and the node ID opposite to the longest edge are displayed in the console.

boundary/modify/repair

Repairs zones by filling all holes associated with free faces. Specify the face zones for the repair operation.

boundary/modify/repair-options/

Contains

boundary/modify/rezone

Moves the selected faces from their current zone into the selected zone, if the selection list contains a zone and one or more faces.

boundary/modify/select-entity

Adds a cell, face, or node to the selection list by entering the name of the entity.

boundary/modify/select-filter

Selects a filter. The possible filters are `off`, `cell`, `face`, `edge`, `node`, `zone`, `position`, `object`, and `size`. If `off` is chosen, then when a selection is made, it is first checked to see if it is a cell, then a face, an edge, and so on. When the `node` filter is used, and if a cell or face is selected, the node closest to the selection point is picked. Thus, the nodes do not have to be displayed, to be picked.

boundary/modify/select-position

Adds a position to the selection list by entering the coordinates of the position.

boundary/modify/select-probe

Selects the probe function. The possible functions are:

- `box` enables the selection of a group of entities within a box, to be used in conjunction with boundary modification functions.

- `label` prints the selection label in the graphics window

- `off` disables the mouse probes.

- `polygon` enables the selection of a group of entities within a polygonal region, to be used in conjunction with boundary modification functions.

- `print` prints the information on the selection in the console window.

- `select` adds the selection to the selection list

boundary/modify/select-visible-entities?

Enables you to select only visible entities (nodes, edges, faces, zones, objects) when the **box** select or **polygon** select options are used. Ensure that the model is zoomed to an appropriate level for correct selection.

Tip:

To quickly revert to the behavior of R16.1 or earlier, enable transparency.

Note:

- If the mesh is not connected, all entities (nodes, edges, faces, zones, objects) will be selected irrespective of whether they are visible or not.

- This visual selection behavior works only on local displays and may generate warning messages when attempting selection on a remote system.

---

boundary/modify/select-zone

Adds a zone to the selection list by entering the zone name or ID.

boundary/modify/show-filter

Shows the current filter.

boundary/modify/show-probe

Shows the current probe function.

boundary/modify/skew

Finds the face with the highest (worst) skewness, selects it in the graphics window, and reports its skewness and zone ID in the console window.

boundary/modify/skew-report-zone

Enables you to select the zone for which you want to report the skewness. You can either specify zone name or zone ID.

boundary/modify/smooth

Uses Laplace smoothing to modify the position of the nodes in the selection list. It moves the selected node to a position computed from an average of its node neighbors. The new position is an average of the neighboring node coordinates and is not reprojected to the discrete surface.

boundary/modify/split-face

Splits two selected faces into four faces.

boundary/modify/swap

Swaps boundary edges (of triangular faces) if the selection list contains edges.

boundary/modify/undo

Undoes the previous operation. When an operation is performed, the reverse operation is stored on the undo stack. For example, a create operation places a delete on the stack, and a delete adds a create operation.

The undo operation requires that the name of the object exist when the action is undone. If the name does not exist, then the undo will fail. You can undo the last few operations, but if many operations are being performed it is recommended that you also save the mesh periodically.

boundary/orient-faces-by-point

Orients the normals based on the specified material point.

boundary/print-info

Prints information about the grid in the text window.

boundary/project-face-zone

Projects nodes on a selected face zone onto a target face zone. Projection can be performed based on normal direction, closest point, or specified direction.

boundary/recover-periodic-surfaces

Restores the periodic relationship between face zones. You will be prompted for the type (rotational or translational), method (semi-automatic, automatic, or manual, depending on the periodicity type) and for face zones. Periodicity information (angle, pivot point, axis of rotation, or translational shift) are read in with the mesh file.

- The `semi-automatic` option prompts you for the source boundary zones and the periodicity information.

- The `automatic` option prompts you for the source boundary zones.

- The `manual` option (available only for rotational periodicity) prompts you for the source and target boundary zones, as well as the periodicity information.

boundary/refine/

Discusses the commands used to refine the boundary mesh.

boundary/refine/auto-refine

Automatically refines a face zone based on proximity. The original face zone is treated as a background mesh. Faces are refined by multiple face splitting passes, so that no face is in close proximity to any face in the current domain.

boundary/refine/clear

Clears all refinement marks from all boundary faces.

boundary/refine/count

Counts the number of faces marked on each boundary zone.

boundary/refine/limits

Prints a report of the minimum and maximum size of each specified zone. This report will also tell you how many faces on each zone have been marked for refinement.

boundary/refine/local-regions/

Enters the local refinement menu.

boundary/refine/local-regions/define

Defines the refinement region according to the specified parameters.

boundary/refine/local-regions/delete

Deletes the specified region.

boundary/refine/local-regions/init

Creates a region encompassing the entire geometry.

boundary/refine/local-regions/list-all-regions

Lists all the refinement regions in the console.

boundary/refine/mark

Marks the faces for refinement.

boundary/refine/refine

Refines the marked faces.

boundary/remesh/

Has a set of commands for remeshing the face zones.

boundary/remesh/clear-marked-faces

Clears the highlighting of the triangles that are marked.

boundary/remesh/coarsen-and-refine

Remeshes (coarsens/refines) the boundary face zones based on the computed size field. Specify the boundary face zones to be remeshed, the boundary edge zones, feature angle, and corner angle. Additionally, specify whether the current boundary face zones should be replaced by the remeshed face zones after the operation is complete.

boundary/remesh/controls/

Enters the edge loop tools text menu.

boundary/remesh/controls/delete-overlapped?

Toggles the deletion of region of overlap of the two surfaces.

boundary/remesh/controls/direction

Specifies the direction of the edge loop projection.

boundary/remesh/controls/intersect/

Enters the intersect control menu.

boundary/remesh/controls/intersect/absolute-tolerance?

Enables you to switch between the use of absolute and relative tolerance. By default, the relative tolerance value is used.

boundary/remesh/controls/intersect/delete-overlap?

Enables/disables the deletion of overlapped edges. It toggles the automatic deletion of region of overlap of the two surfaces. This option is used by while remeshing overlapping zones and retriangulating prisms. By default, this option is enabled.

boundary/remesh/controls/intersect/feature-angle

Specifies the minimum feature angle that should be considered while retriangulating the boundary zones. All the edges in the zone having feature angle greater than the specified feature angle are retained. This option is useful for preserving the shape of the intersecting boundary zones. The default value of feature angle is 40, however, a value in the range of 10–50 degrees is recommended. A large value may distort the shape of the intersecting boundary zones.

boundary/remesh/controls/intersect/ignore-parallel-faces?

Default is yes. If there are close-to-parallel faces, set to no to separate the zones and avoid creating an intersection loop.

boundary/remesh/controls/intersect/join-match-angle

Specifies the allowed maximum angle between the normals of the two overlapping surfaces to be joined. This parameter is used to control the size of the join region.

boundary/remesh/controls/intersect/join-project-angle

Specifies the allowed maximum angle between the face normal and the project direction for the overlapping surfaces to be joined. This parameter is used to control the size of the join region.

boundary/remesh/controls/intersect/refine-region?

Enables you to refine the regions that are modified during the intersect operations. It toggles the refinement of the intersecting regions after performing any of the intersection operation.

This operation improves the quality of the resulting mesh, however, this option is disabled by default.

boundary/remesh/controls/intersect/remesh-post-intersection?

Used to enable or disable automatic post-remesh operation after any connect operation (join, intersect, or stitch).

boundary/remesh/controls/intersect/retri-improve?

Enables you to improve the mesh. After performing any intersection operation, the slivers are removed along the curve of intersection, Laplace smoothing is performed, and followed by the edge swapping. Laplace smoothing is also performed for `insert-edge-zone`, `remesh-overlapped-zones`, and `prism-retriangulation` options. Smoothing is performed again. The smooth-swap operations can be controlled by changing the various defaults such as swapping iterations, smoothing iterations, etc.

boundary/remesh/controls/intersect/separate?

Enables the automatic separation of intersected zones.

boundary/remesh/controls/intersect/stitch-preserve?

Indicates that shape of the first zone specified is to be preserved. This option is enabled by default.

boundary/remesh/controls/intersect/tolerance

Specifies the tolerance value for the intersect operations.

boundary/remesh/controls/intersect/within-tolerance?

Performs the intersection operation only within the specified tolerance value. It is useful only for the **Intersect** option.

boundary/remesh/controls/project-method

Specifies the method for projecting edge loops.

boundary/remesh/controls/proximity-local-search?

Includes the selected face for proximity calculation.

boundary/remesh/controls/quadratic-recon?

Enables/disables quadratic reconstruction of edge loops.

boundary/remesh/controls/remesh-method

Specifies the method to be used for the node distribution on the edge loop.

boundary/remesh/controls/spacing

Sets the node spacing for the edge loop.

boundary/remesh/controls/tolerance

Sets the tolerance for determining if two edges intersect.

boundary/remesh/create-all-intrst-loops

Creates edge loop of intersection for all boundary zones in current domain.

boundary/remesh/create-edge-loops

Creates edge loops for a specified face zone, based on feature angle.

boundary/remesh/create-intersect-loop

Creates an interior edge loop at the intersection between two adjacent face zones. Edges created in this way will not be remeshed by default.

boundary/remesh/create-join-loop

Creates edge loop on boundary of the region of overlap of two surfaces.

boundary/remesh/create-stitch-loop

Creates edge loops for connecting two surfaces along their free edges.

boundary/remesh/delete-overlapped-edges

Deletes edges that overlap selected edge loops.

boundary/remesh/faceted-stitch-zones

Performs the faceted stitching of zones.

boundary/remesh/insert-edge-zone

Inserts an edge zone into a triangulated boundary face zone.

boundary/remesh/intersect-all-face-zones

Remeshes all the intersecting face zones.

After the intersect operation, remesh is called automatically. To disable the post-remesh operation, use the text command:

/boundary/remesh/controls/intersect/remesh-post-intersection? **no**

boundary/remesh/intersect-face-zones

Remeshes two intersecting face zones so that they become conformal.

After the intersect operation, remesh is called automatically. To disable the post-remesh operation, use the text command:

/boundary/remesh/controls/intersect/remesh-post-intersection? **no**

boundary/remesh/join-all-face-zones

Connects all overlapping face zones using the join operation.

After the join operation, remesh is called automatically. To disable the post-remesh operation, use the text command:

`/boundary/remesh/controls/intersect/remesh-post-intersection?` **no**

boundary/remesh/join-face-zones

Connects two overlapping faces.

After the join operation, remesh is called automatically. To disable the post-remesh operation, use the text command:

`/boundary/remesh/controls/intersect/remesh-post-intersection?` **no**

boundary/remesh/mark-intersecting-faces

Highlights the triangles in the neighborhood of the line of intersection.

boundary/remesh/mark-join-faces

Highlights the triangles in the neighborhood of the join edge loop.

boundary/remesh/mark-stitch-faces

Highlights the triangles in the neighborhood of the stitch edge loop.

boundary/remesh/remesh-face-zone

Remeshes a specified face zone by automatically extracting edge loops. If edge loops are present in the current domain (for example, if they were created using the `create-edge-loops` command), they are used to remesh the specified face zone.

boundary/remesh/remesh-constant-size

Remeshes the specified face zones to a constant triangle size while maintaining conformity with adjacent zones. Specify the boundary face zones to be remeshed, the boundary edge zones, feature angle, corner angle, and the constant size. Additionally, specify whether the current boundary face zones should be replaced by the remeshed face zones after the operation is complete.

boundary/remesh/remesh-face-zones-conformally

Remeshes face zones using the current size function and keeping a conformal interface between them. If no size function is defined, an error message will be generated.

This command will prompt for:

- `Boundary Face Zones`

- `Boundary Edge Zones`

- `feature angle` – used to determine the minimum angle between features that will be preserved during remeshing

- `corner angle` – used to specify the minimum angle between feature edges that will be preserved

- `Replace Face Zone?` – (default is Yes) the remeshed face zone(s) will take the name and -id of the original zones, and the original face zone(s) will have "orig" appended to their name. If No, the remeshed face zone(s) will have "retri" added postfix.

Note:

Periodic face zones cannot be remeshed using this command.

---

boundary/remesh/remesh-overlapping-zones

Remeshes overlapping face zones. The non-overlapping region is remeshed using the edge loops created from the overlapping face zones.

boundary/remesh/size-functions/

Enters the size functions menu where you can define size functions for controlling mesh size distribution.

boundary/remesh/size-functions/compute

Computes the size field based on the defined parameters.

boundary/remesh/size-functions/contours/

Contains options for displaying contours of size functions.

boundary/remesh/size-functions/contours/draw

Displays contours in the graphics window. Compute the size field using `/size-functions/compute` or read in a size field file prior to displaying the contours of size.

boundary/remesh/size-functions/contours/set/refine-facets?

Enables you to specify smaller facets if the original are too large. Default is `no`.

boundary/remesh/size-functions/create

Defines the size function based on the specified parameters.

boundary/remesh/size-functions/create-defaults

Creates default size functions based on face and edge curvature and proximity.

boundary/remesh/size-functions/delete

Deletes the specified size function or the current size field.

`boundary/remesh/size-functions/`delete-all

Deletes all the defined size functions.

`boundary/remesh/size-functions/`disable-periodicity-filter

Removes periodicity from the size field.

`boundary/remesh/size-functions/`list

Lists all the defined size functions and the parameter values defined.

`boundary/remesh/size-functions/`reset-global-controls

Resets the global controls to their default values.

`boundary/remesh/size-functions/`set-global-controls

Sets the values for the global minimum and maximum size, and the growth rate.

`boundary/remesh/size-functions/`set-periodicity-filter

Applies periodicity to the size field by selecting one source face zone.

Note:

Ensure that periodicity is previously defined.

Only rotational periodicity is supported, translational periodicity is not supported currently.

`boundary/remesh/size-functions/`set-prox-gap-tolerance

Sets the tolerance relative to minimum size to take gaps into account. Gaps whose thickness is less than the global minimum size multiplied by this factor will not be regarded as a proximity gap.

`boundary/remesh/size-functions/`set-scaling-filter

Specifies the scale factor, and minimum and maximum size values to filter the size output from the size field.

`boundary/remesh/size-functions/`triangulate-quad-faces?

Identifies the zones comprising non-triangular elements and uses a triangulated copy of these zones for computing the size functions.

`boundary/remesh/size-functions/`use-cad-imported-curvature?

Enables/disables curvature data from the nodes of the CAD facets.

`boundary/remesh/`stitch-all-face-zones

Connects (stitches) all the face zones along the free edges.

After the stitch operation, remesh is called automatically. To disable the post-remesh operation, use the text command:

`/boundary/remesh/controls/intersect/remesh-post-intersection?` **no**

boundary/remesh/stitch-face-zones

Connects two surfaces along their free edges.

After the stitch operation, remesh is called automatically. To disable the post-remesh operation, use the text command:

`/boundary/remesh/controls/intersect/remesh-post-intersection?` **no**

boundary/remesh/stitch-with-preserve-boundary

Connects (stitches) a zone to another which is connected to an existing volume mesh, while preserving the boundary of the zones connected to the volume mesh. Specify a list of boundary zones to be preserved, a list of the boundary zones to be connected to each of these zones, and the tolerance value.

After the stitch operation, remesh is called automatically. To disable the post-remesh operation, use the text command:

`/boundary/remesh/controls/intersect/remesh-post-intersection?` **no**

> **Note:**
>
> This command will not work for overlapping or partially overlapping face zones.

boundary/remesh/triangulate

Triangulates quad zones.

boundary/reset-element-type

Resets the element type (mixed, tri, or quad) of a boundary zone. If you have separated a mixed (tri and quad) face zone into one tri face zone and one quad face zone, for example, each of these will be identified as a "mixed" zone. Resetting the element type for each of these new zones will identify them as, respectively, a triangular zone and a quadrilateral zone.

boundary/resolve-face-intersection

Resolves self intersection on manifold surface meshes.

boundary/scale-nodes

Applies a scaling factor to all node coordinates. You can use this command to change the units of the grid.

boundary/separate/

Contains options for separating face zones.

`boundary/separate/local-regions/`

Enters the local refinement menu.

    `boundary/separate/local-regions/define`

    Enables you to define the local region.

    `boundary/separate/local-regions/delete`

    Deletes the specified local region.

    `boundary/separate/local-regions/init`

    Creates a region encompassing the entire geometry.

    `boundary/separate/local-regions/list-all-regions`

    Lists all the local regions defined.

`boundary/separate/mark-faces-in-region`

Marks the faces that are contained in a specified local refinement region.

`boundary/separate/sep-face-zone-by-angle`

Separates a boundary face zone based on significant angle.

`boundary/separate/sep-face-zone-by-cnbor`

Separates a boundary/interior face zone based on its cell neighbors.

`boundary/separate/sep-face-zone-by-mark`

Separates a boundary face zone by moving marked faces to a new zone.

`boundary/separate/sep-face-zone-by-region`

Separates a boundary face zone based on contiguous regions.

`boundary/separate/sep-face-zone-by-seed`

Separates a boundary face zone by defining a seed face on the surface.

`boundary/separate/sep-face-zone-by-seed-angle`

Separates faces connected to the seed face, whose normal fall within the specified cone.

`boundary/separate/sep-face-zone-by-shape`

Separates a boundary face zone based on the shape of the faces (triangular or quadrilateral).

`boundary/set-periodicity`

Defines the periodicity parameters. You will be prompted for the type of periodicity (rotational or translational). For rotational periodicity, you will be prompted for the angle and axis of rotation parameters. For translational periodicity, you will be prompted for the shift vector components.

`boundary/slit-boundary-face`

Slits a boundary face zone by duplicating all faces and nodes, except those nodes that are located at the edges of the boundary zone. A displacement can be specified to provide thickness to the boundary. The slit command only works when it is possible to move from face to face using the connectivity provided by the cells.

You should slit the boundary face *after* you generate the volume mesh so that cells will not be placed inside the gap. There may be some inaccuracies when you graphically display solution data for a mesh with a slit boundary in Ansys Fluent.

`boundary/smooth-marked-faces`

Smooths the marked faces.

`boundary/unmark-selected-faces`

Unmarks the marked selected faces.

`boundary/wrapper/`

Enters the surface wrapper menu.

> **Note:**
>
> This menu is no longer supported, and will be removed in a future release.

`boundary/wrapper/delete-all-cells?`

Deletes the Cartesian mesh. This command is available only after initializing the Cartesian grid.

`boundary/wrapper/imprint-edges?`

Imprints the wrapper surface on recovered feature edges that you recover. This command is available only after creating the wrapper surface.

`boundary/wrapper/initialize`

Creates a Cartesian mesh of the specified parameters.

`boundary/wrapper/local-regions/`

Enters the local refinement menu.

`boundary/wrapper/local-regions/define`

Enables you to define the local refinement region.

`boundary/wrapper/local-regions/`delete

Deletes the specified refinement region.

`boundary/wrapper/local-regions/`init

Creates a region encompassing the entire geometry.

`boundary/wrapper/local-regions/`list-all-regions

Lists all the refinement regions in the console.

`boundary/wrapper/local-regions/`refine

Refines the specified region according to the refinement parameters specified.

`boundary/wrapper/`post-improve/

Enters the wrapper surface improvement options menu.

`boundary/wrapper/post-improve/`auto-post-improve

Improves the wrapper surface using a pre-defined sequence of operations.

`boundary/wrapper/post-improve/`auto-post-wrap

Performs a pre-defined sequence of post-wrapping operations on the wrapper surface.

`boundary/wrapper/post-improve/`coarsen-wrapper-surf

Coarsens the wrapper surface.

`boundary/wrapper/post-improve/`filterout-far-features

Deletes feature edges beyond the specified distance from the wrapper surface.

`boundary/wrapper/post-improve/`imprint-geom-surf

Imprints the geometry threads on wrapper (manual zone recovery).

`boundary/wrapper/post-improve/`improve

Improves the wrapper surface quality based on skewness, size change, or aspect ratio.

`boundary/wrapper/post-improve/`inflate-thin-regions

Pushes apart the overlapping faces in thin regions.

`boundary/wrapper/post-improve/`post-single-surface

Cleans up unmerged island regions after recovering the single surface.

`boundary/wrapper/post-improve/`recover-single-surface

Recovers thin surfaces as a single surface after wrapping.

`boundary/wrapper/post-improve/`recover-zone

Separates the wrapper surface into zones based on the original geometry.

`boundary/wrapper/post-improve/`remove-crossover-config

Removes crossover configurations.

`boundary/wrapper/post-improve/`remove-duplicated-nodes

Removes the duplicate nodes on the wrapper surface.

`boundary/wrapper/post-improve/`rename-wrapper-zones

Renames the wrapper zones by specifying an appropriate prefix instead of the default prefix (**wrap-**).

`boundary/wrapper/post-improve/`resolve-nonmanifoldness

Resolves non-manifold configurations on the wrapper surface.

`boundary/wrapper/post-improve/`resolve-self-intersection

Removes the self intersecting faces.

`boundary/wrapper/post-improve/`rezone

Smooths the zones separated from the wrapper surface for better representation of the geometry.

`boundary/wrapper/post-improve/`smooth-folded-faces

Smooths the folded faces on the wrapper surface.

`boundary/wrapper/post-improve/`smooth-wrapper-surf

Smooths the wrapper surface.

`boundary/wrapper/post-improve/`swap-wrapper-surf

Swaps the nodes of the wrapper surface to improve its quality.

`boundary/wrapper/`pre-smooth?

Enables/disables smoothing of nodes during wrapping. This command is available only after initializing the Cartesian grid.

`boundary/wrapper/`region/

Enters the regions menu. This command is available only after initializing the Cartesian grid.

`boundary/wrapper/region/`draw-holes

Draws the holes detected.

boundary/wrapper/region/delete-interface

Deletes the existing interface.

boundary/wrapper/region/extract-enclosing-region

Extracts the interface for the region enclosing the specified point.

boundary/wrapper/region/extract-interface

Extracts the interfaces for the specified regions.

boundary/wrapper/region/fix-holes

Fixes the specified hole(s).

boundary/wrapper/region/list-holes

Lists the existing holes.

boundary/wrapper/region/list-interfaces

Lists the existing interfaces.

boundary/wrapper/region/list-regions

Lists the regions created during the wrapper initialization.

boundary/wrapper/region/merge-interior-regions

Merges all the interior regions. After using this command, two regions will remain, the exterior and the merged interior region.

boundary/wrapper/region/merge-regions

Merges the specified regions.

boundary/wrapper/region/modify-region-holes

Enables you to fix or open holes related to the specified region.

boundary/wrapper/region/open-holes

Opens the specified hole(s).

boundary/wrapper/region/refine

Refines the Cartesian grid based on the zone specific sizes and local size functions. This command is available only after initializing Cartesian grid.

boundary/wrapper/region/refine-enclosing-region

Refines the region enclosing the specified point.

boundary/wrapper/region/refine-region

Refines the specified region.

boundary/wrapper/region/refine-zone-cells

Refines the cells associated with the specified boundary zone. This command is available only after initializing the Cartesian grid.

boundary/wrapper/region/update-regions

Updates the regions to account for the changes made to the original geometry (during manual hole fixing).

boundary/wrapper/region/wrap-enclosing-region

Generates the wrapper surface for the region enclosing the specified point.

boundary/wrapper/region/wrap-interface

Generates the wrapper surface for the specified interface.

boundary/wrapper/region/wrap-region

Generates the wrapper surface for the specified region. This command is available only after initializing the Cartesian grid.

boundary/wrapper/region/wrapper-region-at-location

Reports the region at the specified location.

boundary/wrapper/set/

Enters the menu to set cell parameters.

boundary/wrapper/set/auto-draw-sizes

Enables you to draw zone-specific sizes.

boundary/wrapper/set/clear-size

Clears all the zone specific size parameters.

boundary/wrapper/set/curvature-factor

Enables you to modify curvature size function sensitivity of a wrapper.

boundary/wrapper/set/curvature?

Enables you to enable or disable curvature size function.

boundary/wrapper/set/default-face-size

Enables you to specify default face size for Cartesian grid.

`boundary/wrapper/set/`feature-threshold

Enables you to specify critical range within which the nodes of the wrapper will be projected onto the feature edges.

`boundary/wrapper/set/`ignore-feature-skewness

Specifies the critical skewness to be considered for ignoring feature lines.

`boundary/wrapper/set/`ignore-self-proximity?

Enables/disables the self-proximity calculation during the refinement.

`boundary/wrapper/set/`list-size

Lists the current zone specific sizes of the domain.

`boundary/wrapper/set/`local-size-function

Enables you to set local size functions.

`boundary/wrapper/set/`max-refine-level

Specifies the maximum refinement level permitted.

`boundary/wrapper/set/`maximum-size-level

Specifies the refinement level of the largest cell in the Cartesian grid.

`boundary/wrapper/set/`minimum-proximity-gap

Specifies the minimum proximity gap within which the proximity will be ignored.

`boundary/wrapper/set/`minimum-size-level

Specifies the refinement level of the smallest cell in the Cartesian grid.

`boundary/wrapper/set/`number-of-size-boxes

Controls the number of boxes to display, when using **Draw Sizes** button in the **Face Size** tab.

`boundary/wrapper/set/`proximity-factor

Enables you to modify proximity size function sensitivity of a wrapper.

`boundary/wrapper/set/`proximity?

Enables/disables proximity size function.

`boundary/wrapper/set/`read-local-sizes

Reads the zone specific sizes stated in a file.

`boundary/wrapper/set/`refinement-buffer-layers

Specifies the number of additional cell layers that you want to refine.

`boundary/wrapper/set/`relative-island-count

Specifies a critical cell count of noise zones in a zone separation.

`boundary/wrapper/set/`volume-marker

Offers a volume marker manipulation for holes.

`boundary/wrapper/set/`write-local-sizes

Writes the zone specific sizes in a file.

`boundary/wrapper/set/`zone-specific-size

Enables you to specify zone specific sizes.

Chapter 2: `cad-assemblies/`

`cad-assemblies/add-prefix`

Enables you to add a prefix to the selected entities. Specify the path for the entities and the prefix to be added.

`cad-assemblies/add-to-object`

Enables you to add the selected CAD entities to an existing object. Specify the path for the entities to be added and select the object to be modified.

`cad-assemblies/create-objects`

Enables you to create new geometry/mesh objects for the selected entities. Specify the path for the entities and if required, choose to create one object per CAD entity selected and/or retain the CAD zone granularity for object creation. By default, a single object will be created for all entities selected and the CAD zone granularity will not be retained. Specify the object name (if applicable), object type (`geom` or `mesh`), and cell zone type (`dead`, `fluid`, or `solid`).

`cad-assemblies/delete-cad-assemblies`

Deletes all the CAD assemblies data.

`cad-assemblies/draw`

Displays the selected CAD entities.

`cad-assemblies/draw-options/`

Contains additional options for displaying CAD entities.

`cad-assemblies/draw-options/add-to-graphics`

Adds the selected entities to the display in the graphics window.

`cad-assemblies/draw-options/draw-unlabelled-zones`

Displays the unlabeled zones for the selected entities in the graphics window.

`cad-assemblies/draw-options/remove-from-graphics`

Removes the selected entities from the display in the graphics window.

`cad-assemblies/extract-edge-zones`

Enables you to extract the feature edge zone for the selected entities. Specify the path for the CAD entities and the feature angle.

cad-assemblies/labels/

Contains options for displaying and managing labels.

cad-assemblies/labels/add-to-graphics

Adds the selected labels to the display in the graphics window.

cad-assemblies/labels/delete

Deletes the selected labels.

cad-assemblies/labels/draw

Displays the selected labels.

cad-assemblies/labels/remove-from-graphics

Removes the selected labels from the display in the graphics window.

cad-assemblies/labels/rename

Enables you to rename the selected labels. Specify the path for the labels and the new name. For multiple selections, the specified name will be used, with a suitable index as suffix. For example, specifying a new label name `wall` will result in entities `wall.1`, `wall.2`, etc.

cad-assemblies/manage-state/

Contains options for setting the CAD entity state.

cad-assemblies/manage-state/suppress

Suppresses the selected CAD entities.

cad-assemblies/manage-state/unlock

Unlocks the selected CAD entities.

cad-assemblies/manage-state/unsuppress

Unsuppresses the selected CAD entities.

cad-assemblies/rename

Enables you to rename the selected entities. Specify the path for the entities and the new name. For multiple entities, the specified name will be used, with a suitable index as suffix. For example, specifying a new name `wall` will result in entities `wall.1`, `wall.2`, etc.

cad-assemblies/replace-object

Enables you to replace an object with the selected CAD entities. Specify the path for the entities to be added and select the object to be modified.

cad-assemblies/update-cad-assemblies

Reimports the selected CAD entities using new parameters specified in the `update-options/` menu.

cad-assemblies/update-options/

Contains options for updating the CAD entities on reimport.

cad-assemblies/update-options/import-edge-zones?

Enables you to import edge zones from the CAD entities on reimport. Specify an appropriate value for feature angle.

cad-assemblies/update-options/one-object-per

Enables you to change the CAD object granularity on reimport.

cad-assemblies/update-options/one-zone-per

Enables you to change the CAD zone granularity on reimport.

cad-assemblies/update-options/tessellation

Enables you to control the tessellation (faceting) during reimport. You can select either `cad-faceting` or `cfd-surface-mesh`.

CAD faceting enables you to control the tessellation based on the CAD faceting tolerance and maximum facet size specified.

CFD Surface Mesh enables you to use a size field file during reimport. If you enter `yes`, specify the size field file to be read. If you do not want to use a size field file, you can obtain conformal faceting based on the underlying curve and surface curvature (using the minimum and maximum facet sizes, and the facet curvature normal angle specified) and edge proximity (using the cells per gap specified). You can also save the size field in a file (size field is computed based on the specified parameters; that is, **Min Size**, **Max Size**, **Curvature Normal Angle**, **Cells Per Gap**).

Chapter 3: `diagnostics/`

`diagnostics/`**face-connectivity/**

Contains options for fixing problems with face connectivity on the specified object face zones or boundary face zones.

`diagnostics/face-connectivity/`**add-label-to-small-neighbors**

Separates island object face zones from all connected neighbors and merges them to the connected neighboring face zone label based on minimum face count specified.

`diagnostics/face-connectivity/`**fix-deviations**

Fixes deviations in the wrapped surface mesh by imprinting edges on the wrapped face zones. Specify the number of imprint iterations and aggressive imprint iterations to be performed.

`diagnostics/face-connectivity/`**fix-duplicate-faces**

Removes duplicate faces.

`diagnostics/face-connectivity/`**fix-free-faces/**

Removes free faces by the method selected. The methods available are:

`diagnostics/face-connectivity/fix-free-faces/`**delete-free-edge-faces**

Removes free faces having the specified number of free edges.

`diagnostics/face-connectivity/fix-free-faces/`**delete-fringes**

Removes free face fringes based on the face count specified.

`diagnostics/face-connectivity/fix-free-faces/`**delete-skewed-faces**

Removes free faces based on the minimum skewness specified.

`diagnostics/face-connectivity/fix-free-faces/`**merge-nodes**

Removes free faces by merging nodes within the tolerance specified.

`diagnostics/face-connectivity/fix-free-faces/`**stitch**

Removes free faces based on the tolerance and number of steps specified.

59

`diagnostics/face-connectivity/`fix-invalid-normals

Fixes invalid normals by smoothing.

> **Note:**
>
> Zone-specific or scoped prism settings should be applied prior to using this command.

`diagnostics/face-connectivity/`fix-islands

Deletes groups of island faces based on the absolute face count specified.

`diagnostics/face-connectivity/`fix-multi-faces

Fixes multiply connected faces by a combination of deleting face fringes, overlapping faces, and disconnected faces. Specify the maximum number of fringe faces, overlapping faces, and multiply connected edges, respectively.

`diagnostics/face-connectivity/`fix-point-contacts

Fixes non-manifold configurations by removing point contacts.

`diagnostics/face-connectivity/`fix-self-intersections

Fixes self intersecting or folded faces. For fixing folded faces by smoothing, specify whether features should be imprinted.

`diagnostics/face-connectivity/`fix-slivers

Fixes faces based on skewness and height criteria. Height is the perpendicular distance between the longest edge of the triangle and the opposite node.

`diagnostics/face-connectivity/`fix-spikes

Fixes spiked faces based on the spike angle specified.

`diagnostics/face-connectivity/`fix-steps

Fixes step configurations by smoothing or collapsing faces based on the angle and step width specified.

`diagnostics/face-connectivity/`remove-label-from-small-neighbors

Removes disconnected island object face zone labels by merging the specified island object face zones to the connected neighboring face zone label based on minimum face count specified.

`diagnostics/`quality/

Contains options for fixing problems related to surface mesh quality on the specified object face zones or boundary face zones.

`diagnostics/quality/`collapse

Collapses bad quality faces based on area or skewness. For collapsing based on face area, specify the maximum face area and relative maximum area. For collapsing based on face skewness, specify the minimum skewness and feature angle. Additionally, specify the number of iterations and whether the boundary should be preserved.

`diagnostics/quality/`delaunay-swap

Improves the surface mesh by swapping based on the minimum skewness value and feature angle specified. Additionally, specify the number of iterations and whether the boundary should be preserved.

`diagnostics/quality/`general-improve

Improves the surface mesh based on aspect ratio, size change, or skewness. Specify the minimum quality value, feature angle, number of iterations, and whether the boundary should be preserved.

`diagnostics/quality/`smooth

Improves the surface mesh by smoothing. Specify the number of smoothing iterations and whether the boundary should be preserved.

Chapter 4: `display/`

`display/`**all-grid**

Displays the grid according to the currently set parameters.

`display/`**annotate**

Adds 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.

`display/`**boundary-cells**

Displays boundary cells attached to the specified face zones.

`display/`**boundary-grid**

Displays only boundary zones according to the currently set parameters.

`display/`**center-view-on**

Sets the camera target to be the center (centroid) of an entity.

`display/`**clear**

Clears the active graphics window. This option is useful when you redo an overlay.

`display/`**clear-annotation**

Removes all annotations and attachment lines from the active graphics window.

`display/`**draw-cells-using-faces**

Draws cells that are neighbors for the selected faces.

`display/`**draw-cells-using-nodes**

Draws cells that are connected to the selected nodes.

`display/`**draw-face-zones-using-entities**

Draws cells that are connected to the selected entities.

`display/`**draw-zones**

Draws the boundary/cell zones using the zone ID specified as input.

display/objects/

Contains commands for displaying objects.

display/objects/display-neighborhood

Displays the objects that are in the neighborhood of the selected object. The neighboring objects have to be in contact, or intersecting the selected object.

display/objects/display-similar-area

Displays the objects with similar area to the selected object area.

display/objects/explode

Explodes the objects in the geometry. (This command is valid only when the geometry is an assembled mode.)

display/objects/hide-objects

Hides the selected objects in the display.

display/objects/implode

Implodes or assembles the objects in the geometry. (This command is available only when the geometry is an exploded mode.)

display/objects/isolate-objects

Displays only the selected objects.

display/objects/make-transparent

Makes the geometry transparent so that internal objects are visible. This command works as a toggle undoing the transparency of the previously selected objects.

display/objects/select-all-visible

Selects all the visible objects in the graphics window.

display/objects/show-all

Unhides all the objects in the geometry and displays them.

display/objects/toggle-color-mode

Toggles the colors of the geometry. In one mode geometry is colored object-wise while in the other mode it is colored zone-wise.

display/objects/toggle-color-palette

Toggles the color palette of the geometry.

display/redisplay

Redraws the grid in the graphics window.

display/save-picture

Saves a picture file of the active graphics window.

display/set-grid/

Contains options controlling the display of the grid.

display/set-grid/all-cells?

Enables/disables the display of all cells.

display/set-grid/all-faces?

Enables/disables the display of all faces.

display/set-grid/all-nodes?

Enables/disables the display of all nodes.

display/set-grid/cell-quality

Sets the lower and upper bounds of quality for cells to be displayed. Only cells with a quality measure value (for example, skewness) within the specified range will be displayed.

display/set-grid/default

Resets the grid display parameters to their default values.

display/set-grid/face-quality

Sets the lower and upper bounds of quality for faces to be displayed. Only faces with a quality measure value (for example, skewness) within the specified range will be displayed.

display/set-grid/free?

Enables/disables the drawing of faces/nodes that have no neighboring face on at least one edge.

display/set-grid/label-alignment

Sets the alignment of labels that appear in the graphics window. By default, the label is centered on the node, cell, and so on, to which the label refers. You can specify `*`, `^`, `v`, `<`, `>` for center, top, bottom, left, or right. You can also combine symbols—for example, `"*v"` for bottom center.

display/set-grid/label-font

Sets the label font. By default, all labels appear in "sans serif" font. Some other choices are roman, typewriter, and stroked.

display/set-grid/label-scale

Scales the size of the label.

display/set-grid/labels?

Enables/disables the display of labels.

display/set-grid/left-handed?

Enables/disables the display of left-handed faces.

display/set-grid/list

Lists all the grid display settings.

display/set-grid/marked?

Enables/disables the display of marked nodes.

display/set-grid/multi?

Enables/disables the display of those faces/nodes that have more than one neighboring face on an edge.

display/set-grid/neighborhood

Sets the x, y, and z range to be within a specified neighborhood of a specified grid object.

display/set-grid/node-size

Sets the node symbol scaling factor.

display/set-grid/node-symbol

Specifies the node symbol.

display/set-grid/normal-scale

Sets the scale factor for face normals.

display/set-grid/normals?

Enables/disables the display of face normals.

display/set-grid/refine?

Enables/disables the display of those faces that have been marked for refinement.

display/set-grid/tagged?

Enables/disables the display of tagged nodes.

display/set-grid/unmeshed?

Enables/disables the display of nodes and faces that have not been meshed.

`display/set-grid/`unused?

Enables/disables the display of unused nodes.

`display/set-grid/`x-range

Limits the display of grid objects to the specified x-range.

`display/set-grid/`y-range

Limits the display of grid objects to the specified y-range.

`display/set-grid/`z-range

Limits the display of grid objects to the specified z-range.

`display/`set-list-tree-separator

Sets the separator character to be used to determine the common prefix for items listed in the selection lists, when the tree view is used.

`display/`set/

Enables you to enter the set menu to set the display parameters.

`display/set/`colors/

Enables you to enter the colors options menu.

`display/set/colors/`axis-faces

Sets the color of axisymmetric faces.

`display/set/colors/`background

Sets the background (window) color.

`display/set/colors/`color-by-type?

Enables you to specify that the entities should be colored by their type or ID.

`display/set/colors/`color-by-partition?

Enables you to view the partitions by color. This command applies to parallel processing.

`display/set/colors/`far-field-faces

Sets the color of far field faces.

`display/set/colors/`foreground

Sets the foreground (text and window frame) color.

`display/set/colors/`free-surface-faces

Sets the color of free surface faces.

display/set/colors/graphics-color-theme

Sets the color theme for the graphics window. The color options (`black`, `white`, `gray-gradient`, or `workbench`) are for the background display, but changing the theme also changes the default colors for items that display in the graphics windows, like faces and edges.

display/set/colors/highlight-color

Sets the highlight color.

display/set/colors/inlet-faces

Sets the color of the inlet faces.

display/set/colors/interface-faces

Sets the color of grid interface faces.

display/set/colors/interior-faces

Sets the color of the interior faces.

display/set/colors/internal-faces

Sets the color of the internal interface faces.

display/set/colors/list

Lists the colors available for the selected zone type.

display/set/colors/outlet-faces

Sets the color of the outlet faces.

display/set/colors/overset-faces

Sets the color of the overset faces.

display/set/colors/periodic-faces

Sets the color of periodic faces.

display/set/colors/rans-les-interface-faces

Sets the color of RANS/LES interface faces.

display/set/colors/reset-colors

Resets the individual grid surface colors to the defaults.

display/set/colors/reset-user-colors

Resets individual grid surface colors to the defaults.

`display/set/colors/`show-user-colors

Lists the current defined user colors.

`display/set/colors/`skip-label

Sets the number of labels to be skipped in the colormap scale.

`display/set/colors/`surface

Sets the color of surfaces.

`display/set/colors/`symmetry-faces

Sets the color of symmetric faces.

`display/set/colors/`traction-faces

Sets the color for traction faces.

`display/set/colors/`user-color

Enables you to change the color for the specified zone.

`display/set/colors/`wall-faces

Sets color for wall faces.

`display/set/`edges?

Enables/disables the display of face/cell edges.

`display/set/`filled-grid?

Enables/disables the filled grid option. When a grid is not filled, only its outline is drawn.

`display/set/`lights/

Enters the lights menu.

`display/set/lights/`headlight-on?

Turns the light that moves with the camera on or off. This is controlled automatically by default.

`display/set/lights/`lighting-interpolation

Sets the lighting interpolation method to be used. You can choose `automatic`, `flat`, `gouraud`, or `phong`. "Automatic" automatically picks the best lighting method for the display in the graphics window. Flat is the most basic method, and the others are more sophisticated and provide smoother gradations of color.

`display/set/lights/`lights-on?

Enables/disables the display of all lights.

display/set/lights/set-ambient-color

Sets the ambient color for the scene. The ambient color is the background light color in scene.

display/set/lights/set-light

Adds or modifies a directional, colored light.

display/set/line-weight

Sets the line width factor for the window.

display/set/native-display-defaults

Resets graphics window parameters to optimal settings for a local display.

Used after setting parameters for a remote display with `remote-display-defaults`.

display/set/overlays?

Turns overlays on and off.

display/set/picture/

Saves a hardcopy file of the active graphics window.

display/set/picture/color-mode/

Contains the available color modes.

display/set/picture/color-mode/color

Selects full color and plots the hardcopy in color.

display/set/picture/color-mode/gray-scale

Selects gray scale (that is, various shades of gray) and converts color to gray-scale for hardcopy.

display/set/picture/color-mode/list

Displays the current hardcopy color mode.

display/set/picture/color-mode/mono-chrome

Selects color to monochrome (black and white) for hardcopy.

display/set/picture/dpi

Specifies the resolution in dots per inch for EPS and PostScript files.

display/set/picture/driver/

Contains the available hardcopy formats.

display/set/picture/driver/dump-window

Sets the command to dump a graphics window to a file.

display/set/picture/driver/eps

Sets the Encapsulated PostScript format.

display/set/picture/driver/jpeg

Sets the JPEG image format.

display/set/picture/driver/list

Displays the current hardcopy format.

display/set/picture/driver/options

Enables you to set hardcopy options, such as landscape orientation, pen speed, and physical size. The options may be entered on one line if you separate them with commas.

display/set/picture/driver/png

Sets the PNG image format.

display/set/picture/driver/post-format/

Contains commands for setting the PostScript driver format and save files in PS files that can be printed quickly.

display/set/picture/driver/post-format/fast-raster

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

display/set/picture/driver/post-format/raster

Enables the standard raster file.

display/set/picture/driver/post-format/rle-raster

Enables a run-length encoded raster file that will be about the same size as the standard raster file, but will print slightly more quickly. This is the default file type.

display/set/picture/driver/post-format/vector

Enables the standard vector file.

display/set/picture/driver/post-script

Sets the PostScript format.

display/set/picture/driver/ppm

Sets the PPM format.

display/set/picture/driver/tiff

Sets the TIFF format.

display/set/picture/driver/vrml

Sets the VRML format.

display/set/picture/invert-background?

Enables/disables the exchange of foreground/background colors for hardcopy files.

jpeg-hardcopy-quality

Controls the size and quality of how JPEG files are saved based on a scale of 0-100, with zero being low quality small files and 100 being high quality larger files.

display/set/picture/landscape?

Toggles between landscape or portrait orientation.

display/set/picture/preview

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

display/set/picture/use-window-resolution?

Disables/enables the use of the current graphics window resolution when saving an image of the graphics window. If disabled, the resolution will be as specified for x-resolution and y-resolution.

display/set/picture/x-resolution

Sets the width of the raster format images in pixels (0 implies that the hardcopy should use the same resolution as the active graphics window).

display/set/picture/y-resolution

Sets the height of the raster format images in pixels (0 implies that the hardcopy should use the same resolution as the active graphics window).

display/set/re-render

Re-renders the current window after modifying the variables in the set menu.

display/set/remote-display-defaults

Adjusts graphics window parameters to optimal settings for a remote display.

Restore parameters for local display using native-display-defaults.

display/set/rendering-options/

Contains the commands that enable you to set options that determine how the scene is rendered.

`display/set/rendering-options/`animation-option/

Enables you to specify the animation option as appropriate.

`display/set/rendering-options/animation-option/`all

Uses a solid-tone shading representation of all geometry during mouse manipulation.

`display/set/rendering-options/animation-option/`wireframe

Uses a wireframe representation of all geometry during mouse manipulation. This is the default option.

`display/set/rendering-options/`auto-spin?

Enables mouse view rotations to continue to spin the display after the button is released.

`display/set/rendering-options/`color-map-alignment

Sets the color bar alignment.

`display/set/rendering-options/`device-info

Prints out information about your graphics driver.

`display/set/rendering-options/`double-buffering?

Enables or disables 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.

`display/set/rendering-options/`driver

Changes the current graphics driver. When enabling graphics display, you have various options: for Linux, the available drivers include `opengl` and `x11`; for Windows, the available drivers include `opengl`, `dx11` (for DirectX 11), and `msw` (for Microsoft Windows). You can also disable the graphics display window by entering `null`. For a comprehensive list of the drivers available to you, press the **Enter** key at the `driver>` prompt.

> **Note:**
>
> For any session that displays graphics in a graphics window and/or saves picture files, having the driver set to `x11`, `msw`, or `null` will cause the rendering / saving speed to be significantly slower.

`display/set/rendering-options/`face-displacement

Sets the face displacement (in Z-buffer units along the camera Z-axis) for the displayed geometry when both faces and edges are displayed simultaneously.

display/set/rendering-options/help-text-color

Sets the color of the help text on the screen. You can select `black`, `default`, or `white`.

display/set/rendering-options/hidden-lines?

Turns hidden line removal on or off. This command is available only when the color scheme is set to `classic`.

> **Note:**
>
> This command (only available when **Graphics Color Theme** is set to **Black**) is deprecated and will be removed at a future release.

display/set/rendering-options/hidden-lines-method/

Enables you to select the hidden line removal algorithm.

display/set/rendering-options/hidden-lines-method/mesh-display-hlr?

Enables you to remove hidden lines for surfaces that are very close together. This option should be used only if the default algorithm does not produce suitable results.

display/set/rendering-options/hidden-lines-method/normal-hlr-algorithm

Is the default hidden line removal algorithm.

> **Note:**
>
> These commands (only available when **Graphics Color Theme** is set to **Black**) are deprecated and will be removed at a future release.

display/set/rendering-options/hidden-surface-method/

Enables you to choose from among the hidden surface removal methods that are supported. These options (listed below) are display hardware dependent.

display/set/rendering-options/hidden-surface-method/hardware-z-buffer

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

display/set/rendering-options/hidden-surface-method/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.

display/set/rendering-options/hidden-surface-method/software-z-buffer

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

display/set/rendering-options/hidden-surface-method/z-sort-only

Is a fast software method, but it is not as accurate as `software-z-buffer`.

display/set/rendering-options/hidden-surfaces?

Enables/disables the display of hidden surfaces.

display/set/rendering-options/outer-face-cull?

Enables/disables the display of outer faces.

display/set/rendering-options/set-rendering-options

Sets the rendering options.

display/set/rendering-options/surface-edge-visibility

Controls whether or not the mesh edges are drawn.

display/set/reset-graphics

Resets the graphics system.

display/set/shrink-factor

Sets shrinkage of both faces and cells. A value of zero indicates no shrinkage, while a value of one would shrink the face or cell to a point.

display/set/styles/

Contains commands for setting the display style for the different types of nodes and faces that can be displayed.

display/set/styles/cell-quality

Indicates cells within the specified cell quality range.

display/set/styles/cell-size

Indicates cells within the specified cell size range.

display/set/styles/face-quality

Indicates faces within the specified face quality range.

display/set/styles/face-size

Indicates faces within the specified face size range.

display/set/styles/free

Indicates free nodes or faces.

display/set/styles/left-handed

Indicates faces that do not follow the right-hand rule with respect to their cell neighbors.

display/set/styles/mark

Indicates marked objects (for expert users).

display/set/styles/multi

Indicates multiply-connected nodes or faces.

display/set/styles/refine

Indicates boundary faces to be refined.

display/set/styles/tag

Indicates tagged objects (for expert users).

display/set/styles/unmeshed

Indicates unmeshed nodes or faces.

display/set/styles/unused

Indicates unused nodes or faces.

display/set/title

Sets the problem title.

display/set/windows/

Enters the windows options menu, which contains commands that enable you to customize the relative positions of sub-windows inside the active graphics window.

The menu structure for the axes, main, scale, text, video, and xy submenus is similar.

display/set/windows/aspect-ratio

Sets the aspect ratio of the active window.

display/set/windows/axes/

Enters the axes window options menu.

display/set/windows/axes/border?

Sets whether or not to draw a border around the axes window.

display/set/windows/axes/bottom

Sets the bottom boundary of the axes window.

display/set/windows/axes/clear

Sets the transparency of the axes window.

display/set/windows/axes/left

Sets the left boundary of the axes window.

display/set/windows/axes/right

Sets the right boundary of the axes window.

display/set/windows/axes/top

Sets the top boundary of the axes window.

display/set/windows/axes/visible?

Controls the visibility of the axes window.

display/set/windows/main/

Enters the main view window options menu.

display/set/windows/main/border?

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

display/set/windows/main/bottom

Sets the bottom boundary of the main viewing window.

display/set/windows/main/left

Sets the left boundary of the main viewing window.

display/set/windows/main/right

Sets the right boundary of the main viewing window.

display/set/windows/main/top

Sets the top boundary of the main viewing window.

display/set/windows/main/visible?

Controls the visibility of the main viewing window.

display/set/windows/scale/

Enters the color scale window options menu.

display/set/windows/scale/border?

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

display/set/windows/scale/bottom

Sets the bottom boundary of the color scale window.

display/set/windows/scale/clear?

Sets the transparency of the color scale window.

display/set/windows/scale/font-size

Sets the font size of the color scale window.

display/set/windows/scale/format

Sets the number format of the color scale window (for example, %0.2e).

display/set/windows/scale/left

Sets the left boundary of the color scale window.

display/set/windows/scale/margin

Sets the margin of the color scale window.

display/set/windows/scale/right

Sets the right boundary of the color scale window.

display/set/windows/scale/top

Sets the top boundary of the color scale window.

display/set/windows/scale/visible?

Controls the visibility of the color scale window.

display/set/windows/text/

Enters the text window options menu.

display/set/windows/text/application?

Shows or hides the application name in the picture.

display/set/windows/text/border?

Sets whether or not to draw a border around the text window.

display/set/windows/text/bottom

Sets the bottom boundary of the text window.

display/set/windows/text/clear?

Enables/disables the transparency of the text window.

display/set/windows/text/company?

Shows or hides the company name in the picture.

display/set/windows/text/date?

Shows or hides the date in the picture.

display/set/windows/text/left

Sets the left boundary of the text window.

display/set/windows/text/right

Sets the right boundary of the text window.

display/set/windows/text/top

Sets the top boundary of the text window.

display/set/windows/text/visible?

Controls the visibility of the text window.

display/set/windows/video/

Contains options for modifying a video. This menu is not relevant for the meshing mode.

display/set/windows/video/background

Sets 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.

display/set/windows/video/color-filter

Sets 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.

display/set/windows/video/foreground

Sets 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.

display/set/windows/video/on?

Enables or disables the video picture settings.

display/set/windows/video/pixel-size

Sets the window size in pixels.

`display/set/windows/`xy/

Enters the XY plot window options menu.

`display/set/windows/xy/`border?

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

`display/set/windows/xy/`bottom

Sets the bottom boundary of the XY plot window.

`display/set/windows/xy/`left

Sets the left boundary of the XY plot window.

`display/set/windows/xy/`right

Sets the right boundary of the XY plot window.

`display/set/windows/xy/`top

Sets the top boundary of the XY plot window.

`display/set/windows/xy/`visible?

Controls the visibility of the XY plot window.

`display/`show-hide-clipping-plane-triad

Toggles the display of the clipping plane's triad.

`display/`update-scene/

Contains commands that enable you to update the scene description.

`display/update-scene/`delete

Deletes the geometry selected using the `select-geometry` command.

`display/update-scene/`display

Displays the geometry selected using the `select-geometry` command.

`display/update-scene/`draw-frame?

Enables/disables the drawing of the bounding frame.

`display/update-scene/`overlays?

Enables/disables the overlays option.

`display/update-scene/`select-geometry

Enables you to select the geometry to be updated.

display/update-scene/set-frame

Enables you to change the frame options.

display/update-scene/transform

Enables you to apply the transformation matrix to the geometry selected using the `select-geometry` command.

display/views/

Enters the view window options menu.

display/views/auto-scale

Scales and centers the current scene without changing its orientation.

display/views/camera/

Contains commands to set the camera options.

display/views/camera/dolly-camera

Enables you to move the camera left, right, up, down, in, and out.

display/views/camera/field

Enables you to set the field of view (width and height) of the scene.

display/views/camera/orbit-camera

Enables you to move the camera around the target. Gives the effect of circling around the target.

display/views/camera/pan-camera

Gives you the effect of sweeping the camera across the scene. The camera remains at its position but its target changes.

display/views/camera/position

Sets the camera position.

display/views/camera/projection

Lets you switch between perspective and orthographic views.

display/views/camera/roll-camera

Lets you adjust the camera up-vector.

display/views/camera/target

Sets the point the camera will look at.

display/views/camera/up-vector

Sets the camera up-vector.

display/views/camera/zoom-camera

Adjusts 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).

display/views/default-view

Resets the view to front and center.

display/views/delete-view

Deletes a particular view from the list of stored views.

display/views/last-view

Returns to the camera position before the last manipulation.

display/views/list-views

Lists all predefined and saved views.

display/views/read-views

Reads views from an external view file.

display/views/restore-view

Sets the current view to one of the stored views.

display/views/save-view

Saves the currently displayed view into the list of stored views.

display/views/write-views

Writes views to an external view file.

display/xy-plot/

Enters the XY plot menu.

display/xy-plot/cell-distribution

Plots a histogram of cell quality.

display/xy-plot/face-distribution

Plots a histogram of face quality.

`display/xy-plot/`file

Enables you to choose a file from which to create an xy plot.

`display/xy-plot/`set/

Enters the set window options menu.

`display/xy-plot/set/`auto-scale?

Sets the range for the x- and y-axis. If auto-scaling is not activated for a particular axis, you will be prompted for the minimum and maximum data values.

`display/xy-plot/set/`background-color

Sets the color of the field within the abscissa and ordinate axes.

`display/xy-plot/set/`file-lines

Sets the parameters for plot lines.

`display/xy-plot/set/`file-markers

Sets the parameters for data markers.

`display/xy-plot/set/`key

Sets the visibility and title of the description box that displays the markers and/or lines with their associated data. The key can be positioned and resized using the left mouse button.

`display/xy-plot/set/`labels

Sets the strings that define the x- and y- axis labels.

`display/xy-plot/set/`lines

Sets the pattern, weight, and color of the plot lines.

`display/xy-plot/set/`log?

Enables/disables the log scaling on the x- and y-axis.

`display/xy-plot/set/`markers

Sets parameters for the data markers.

`display/xy-plot/set/`numbers

Sets the format and precision of the data numbers displayed on the x- and y-axis.

`display/xy-plot/set/`plot-to-file

Enables you to write the xy plot values to a file.

83

display/xy-plot/set/rules

Sets the visibility, line weight and color of the major and minor rules in the x- and y-axis directions.

display/xy-plot/set/windows/

Contains commands that enable you to customize the relative positions of sub-windows inside the active graphics window.

display/xy-plot/set/windows/border?

Draws a border around the sub-window.

display/xy-plot/set/windows/bottom

Sets the bottom boundary of the sub-window.

display/xy-plot/set/windows/left

Sets the left boundary of the sub-window.

display/xy-plot/set/windows/right

Sets the right boundary of the sub-window.

display/xy-plot/set/windows/top

Sets the top boundary of the sub-window.

display/xy-plot/set/windows/visible?

Sets the visibility of the sub-window.

display/xy-plot/set/xy-percent-y?

Enables/disables whether the y-coordinate should be scaled to show a percent of total values being plotted.

display/zones/

Contains commands for displaying zones.

display/zones/display-neighborhood

Displays the zones that are in the neighborhood of the selected zones. The neighboring zones have to be in contact, or intersecting the selected zone.

display/zones/display-similar-area

Displays the zones with similar area to the selected zone area.

display/zones/hide-zones

Hides the selected zones in the display.

`display/zones/`isolate-zones

Displays only the selected zones.

`display/zones/`make-transparent

Makes the geometry transparent so that internal zones are visible. This command works as a toggle undoing the transparency of the previously selected zones.

`display/zones/`select-all-visible

Selects all the visible zones in the graphics window.

`display/zones/`show-all

Unhides all the zones in the geometry and displays them.

`display/zones/`toggle-color-mode

Toggles the colors of the geometry. In one mode geometry is colored object-wise while in the other mode it is colored zone-wise.

`display/zones/`toggle-color-palette

Toggles the color palette of the geometry.

Chapter 5: `exit`

exit

Exits the program.

Chapter 6: `file/`

The user interface commands related to the **File** menu (such as reading files, importing files) and other **Select File** dialog boxes do not work for the dual process build. You need to use the TUI commands instead (for example, `/file/read-mesh`).

Important:

- The host cannot be detached and reattached; once the connection is broken the data is lost. You need to save the data if the machine must be shut down in between.

- All graphics information will be sent over the network, so initially it could take a long time to assemble graphical information (especially if the host and remote server are across continents) but after that the graphics manipulation is fast.

`file/`**append-mesh**

Enables you to append the mesh files. This command is available only after a mesh file has been read in.

Append Rules:

- If zone names and IDs are duplicated, they will be modified and the changes will be reported in the console.

- Domain information will be retained during the file append operation. If domain names are duplicated, they will be modified and the changes will be reported in the console.

- Refinement region information will be retained during the file append operation. If region names are duplicated, they will be modified and the changes will be reported in the console.

- You can append files comprising only edge zones (without face zones).

- Edge-face zone associations will be retained during the file append operation.

- Zone-specific prism parameter information will be retained during the file append operation.

`file/`**append-meshes-by-tmerge**

Enables you to append the mesh files using the **tmerge** utility. This command is available only after a mesh file has been read in.

`file/`**cff-files?**

Answering **yes** will set the Common Fluids Format (CFF) as the default file format for reading and writing case/data files.

`file/`confirm-overwrite?

Controls whether attempts to overwrite existing files require confirmation.

If you do not want Ansys Fluent to ask you for confirmation before it overwrites existing files, you can enter the `file/confirm-overwrite?` text command and answer `no`.

`file/`file-format

Enables/disables the writing of binary files.

`file/`filter-list

Lists the names of the converters that are used to change foreign mesh (while importing mesh files from third-party packages) files.

`file/`filter-options

Enables you to change the extension (such as .cas, .msh, .neu) and arguments used with a specified filter.

For example, if you saved the PATRAN files with a `.NEU` extension instead of `.neu`, you can substitute or add `.NEU` to the extension list. For some filters, one of the arguments will be the dimensionality of the grid.

When you use the `filter-options` command for such a filter, you will see a default dimensionality argument of `-d a`. The dimension will automatically be determined, so you need not substitute `2` or `3` for `a`.

`file/`import/

Enables you to import mesh information generated by some CAD packages (Ansys, I-deas, NASTRAN, PATRAN, and HYPERMESH), as well as mesh information in the CGNS (CFD general notation system) format.

These files are imported using the associated text commands listed here:

`file/import/`ansys-surf-mesh

Enables you to read a Ansys surface mesh file.

`file/import/`ansys-vol-mesh

Enables you to read a Ansys volume mesh file.

`file/import/`cad

Enables you to import CAD files based on the options set.

- To import a single file (default), specify the file path and set up options for extracting features, importing curvature data from CAD, and the length unit.

- To import multiple files, specify the directory path and pattern, and set up options for appending the files, extracting features, importing curvature data from CAD and the length unit.

`file/import/`**`cad-geometry`**

Enables you to import CAD files based on the options set.

- To import a single file (default), specify the file path. Set up options for the length unit, tessellation method, and sizing parameters based on the tessellation method.

- To import multiple files, specify the directory path and pattern. Set up options for appending the files, the length unit, tessellation method, and sizing parameters based on the tessellation method.

`file/import/`**`cad-options/`**

Contains additional options for importing CAD files.

`file/import/cad-options/`**`continue-on-error?`**

Enables you to continue the import of the CAD file(s), despite errors or problems creating the faceting on certain surfaces, or other issues. This option is disabled by default.

`file/import/cad-options/`**`create-cad-assemblies?`**

Enables creating the **CAD Assemblies** tree on CAD import. The CAD Assemblies tree represents the CAD tree as it is presented in the CAD package in which it was created. All sub-assembly levels from the CAD are maintained on import in Fluent Meshing.

For commands specific to the CAD assemblies, refer to

`file/import/cad-options/`**`derive-zone-name-from-object-scope?`**

Enables zones without Named Selections to inherit the object name on import. This option is disabled by default.

`file/import/cad-options/`**`double-connected-face-label`**

Adds the specified label to the name of double-connected face zones (face zones shared by two bodies).

`file/import/cad-options/`**`enclosure-symm-processing?`**

Enables processing of enclosure and symmetry named selections during import. This option is disabled by default. This option is applicable only to Ansys DesignModeler (*.agdb) files.

`file/import/cad-options/`**`extract-features?`**

Enables feature extraction from the CAD model on import. You can choose to disable this, if desired. Specify an appropriate value for feature angle. The default value is 40.

`file/import/cad-options/import-body-names?`

Enables import of Body names from the CAD files. This option is enabled by default.

> **Note:**
>
> Any renaming of Body names in Ansys Mechanical/Ansys Meshing prior to the export of the mechdat/meshdat files is ignored during import. Only original Body names will be imported.

`file/import/cad-options/import-curvature-data-from-CAD?`

Enables importing of the curvature data from the nodes of the CAD facets. You can choose to disable this, if desired.

`file/import/cad-options/import-part-names?`

Enables import of Part names from the CAD file(s). This option is enabled by default.

> **Note:**
>
> Any renaming of Part names in Ansys Mechanical/Ansys Meshing prior to the export of the mechdat/meshdat files is ignored during import. Only original Part names will be imported.

`file/import/cad-options/merge-nodes?`

Enables the merging of geometry object nodes during CAD import. This option is enabled by default.

> **Note:**
>
> This option can be optionally enabled/disabled only when geometry objects are imported using the **CAD Faceting** option for CAD import. Mesh object nodes will always be merged when the **CFD Surface Mesh** is selected for CAD import.

`file/import/cad-options/modify-all-duplicate-names?`

Enables you to modify all duplicate object/zone names by adding incremental integers as suffix. This option is disabled by default.

For example: The CAD file contains multiple parts (or bodies) named **heatshield**.

- With the option disabled (default), the imported zones will be named **heatshield**, **heatshield.1**, **heatshield.2**, etc.

- With the option enabled, the imported zones will be named **heatshield.1**, **heatshield.2**, **heatshield.3**, etc.

`file/import/cad-options/`name-separator-character

Allows you to specify the character used between name fields. Default is ':'.

`file/import/cad-options/`named-selections

Enables you to import **Named Selections** from the CAD file(s), including Named Selections from Ansys DesignModeler, publications from CATIA, and so on. You can additionally choose to ignore import of certain Named Selections based on the pattern specified (for example, `Layer*` to ignore layer Named Selections from CATIA), or by specifying multiple wild cards (for example, `^(Color|Layer|Material).*` to remove color, layer, and material Named Selections from CATIA).

> **Note:**
>
> - Named Selections defined in Ansys Meshing cannot be imported.
>
> - If Named Selections is enabled, then Face named selections will be imported as face zone labels.

`file/import/cad-options/`object-type

Enables the setting of object type on import. The options available are `auto`, `geometry`, and `mesh`. The default setting is `auto` based on the tessellation method selected: geometry objects will be created when the `cad-faceting` method is used, while mesh objects will be created when the `cfd-surface-mesh` method is used.

`file/import/cad-options/`one-face-zone-per

Enables you to create one face zone per body/face/object to be imported.

`file/import/cad-options/`one-object-per

Enables you to create one object per body/part/file/selection to be imported. The default `program-controlled` option allows the software to make the appropriate choice. This option makes a choice between per body and per part based on whether shared topology is off or on, respectively.

> **Note:**
>
> For Ansys ICEM CFD files (*.tin), set the object granularity to one object per **selection**.

`file/import/cad-options/`read-all-cad-in-subdirectories?

When enabled, all files in the specified directory as well as in its subdirectories will be imported. This option is disabled by default.

`file/import/cad-options/`save-PMDB?

Saves a PMDB (*.pmdb) file in the directory containing the CAD files imported. You can use this file to import the same CAD file(s) again with different options set, for a quicker import than the full import. This option is disabled by default.

Note:

Some options will not be available any more once the model is imported from a PMDB file (for example, `enclosure-symm-processing?`), since they are processed before the PMDB file is created.

`file/import/cad-options/`separate-features-by-type?

Enables separation of feature edges based on angle, connectivity, and named selections on import. Edge zone names will have suitable suffixes depending on separation criteria, order of zones, existing zone names and other import options selected.

`file/import/cad-options/`single-connected-edge-label

Adds the specified label to the name of single-connected edge zones (edge zones referenced by a single face).

`file/import/cad-options/`strip-file-name-extension-from-naming?

Removes the extension of the CAD files from the object/face zone names on import. This option is disabled by default.

`file/import/cad-options/`strip-path-prefix-from-names?

Enables you to remove the path prefix from the object/face zone names on import. The default setting is `auto` which removes the path prefix from object/face zone names when the object creation granularity is set to one object per file. You can also explicitly select `yes` or `no`.

`file/import/cad-options/`tessellation

Enables you to control the tessellation (faceting) during file import. You can select either `cad-faceting` or `cfd-surface-mesh`.

CAD faceting enables you to control the tessellation based on the CAD faceting tolerance and maximum facet size specified.

CFD Surface Mesh enables you to use a size field file, (`Use size field file?`). If you enter `yes`, specify the size field file to be read. If you do not want to use a size field file, you can obtain conformal faceting based on the underlying curve and surface curvature (using the minimum and maximum facet sizes, and the facet curvature normal angle specified) and edge proximity (using the cells per gap specified). You can also save a size field in a file (size field is computed based on the specified parameters; that is, **Min Size**, **Max Size**, **Curvature Normal Angle**, **Cells Per Gap**).

`file/import/cad-options/use-collection-names?`

Enables you to use the Named Selections for the object/zone names on import. Select `auto`, `no`, or `yes`. The default selection is `auto` where the Named Selection will be used as the object/zone name, except when the object creation granularity is set to one object per file.

`file/import/cad-options/use-component-names?`

Enables you to add the component (part or assembly) names to the object/zone names on import. Select `auto`, `no`, or `yes`. The default selection is `auto` where the component name will be added to the object/zone name.

`file/import/cad-options/use-part-names?`

Enables you to choose whether to add the part names from the CAD file to the object and zone names on import. The default setting is `auto` which adds the part names to both object and zone names when object creation granularity is set to **body**. When the object creation granularity is set to **part** or **file**, the part names are not added to the zone names, face zone labels, or the region names, by default. You can also explicitly select `yes` or `no`.

`file/import/cad-options/use-part-or-body-names-assuffix-to-named-selections?`

Enables you to modify zone names by using part or body names as suffixes to the Named Selections spanning multiple parts/bodies. This option is enabled by default.

For example: The CAD file contains a Named Selection **effusion** with part (or body) names **id_liner** and **od_liner**.

- With the option enabled (default), the imported zones will be named **effusion:id_liner** and **effusion:od_liner**.

- With the option disabled, the imported zones will be named **effusion.1** and **effusion.2**.

`file/import/cgns-surf-mesh`

Enables you to read a CGNS surface mesh file.

`file/import/cgns-vol-mesh`

Enables you to read a CGNS volume mesh file.

`file/import/fidap-surf-mesh`

Enables you to read a FIDAP surface mesh file.

`file/import/fidap-vol-mesh`

Enables you to read a FIDAP volume mesh file.

`file/import/fl-uns2-mesh`

Enables you to read a Fluent UNS V2 case file.

file/import/fluent-2d-mesh

Enables you to read a 2D mesh into the 3D version.

file/import/gambit-surf-mesh

Enables you to read a GAMBIT surface mesh file.

file/import/gambit-vol-mesh

Enables you to read a GAMBIT volume mesh file.

file/import/hypermesh-surf-mesh

Enables you to read a HYPERMESH surface mesh file.

file/import/hypermesh-vol-mesh

Enables you to read a HYPERMESH volume mesh file.

file/import/ideas-surf-mesh

Enables you to read an I-deas surface mesh file.

file/import/ideas-vol-mesh

Enables you to read an I-deas volume mesh file.

file/import/nastran-surf-mesh

Enables you to read a NASTRAN surface mesh file.

file/import/nastran-vol-mesh

Enables you to read a NASTRAN volume mesh file.

file/import/patran-surf-mesh

Enables you to read a PATRAN surface mesh file.

file/import/patran-vol-mesh

Enables you to read a PATRAN volume mesh file.

file/load-act-tool

Loads the Ansys ACT tool.

file/read-boundary-mesh

Enables you to read a boundary mesh. If the boundary mesh is contained in two or more separate files, you can read them in together and assemble the complete boundary mesh.

96

This option is also convenient if you want to reuse the boundary mesh from a file containing a large volume mesh.

> **Note:**
>
> The naming of face zones can be controlled by Named Selections defined in Ansys Workbench. For details on exporting faceted geometry from Ansys Workbench, refer to the Ansys Workbench Help.

file/read-case

Enables you to read the mesh contained in a case file.

> **Note:**
>
> Cell hierarchy in case files adapted in the solution mode will be lost when they are read in the meshing mode.

Case files containing polyhedral cells can also be read in the meshing mode of Fluent. You can display the polyhedral mesh, perform certain mesh manipulation operations, check the mesh quality, and so on.

file/read-domains

Enables you to read domain files.

Each mesh file written by Fluent has a domain section. A domain file is the domain section of the mesh file and is written as a separate file. It contains a list of node, face, and cell zone IDs that make up each domain in the mesh.

If a domain that is being read already exists in the mesh, a warning message is displayed. Fluent verifies if the zones defining the domains exist in the mesh. If not, it will display a warning message.

file/read-journal

Enables you to read a journal file into the program.

The `read-journal` command always loads the file in the main (that is, top-level) menu, regardless of where you are in the menu hierarchy when you invoke it.

file/read-mesh

Enables you to read a mesh file. You can also use this command to read a Fluent mesh file created with GAMBIT, or to read the mesh available in a Fluent case file.

> **Note:**
>
> Reading a case file as a mesh file will result in loss of boundary condition data as the mesh file does not contain any information on boundary conditions.

Case files containing polyhedral cells can also be read in the meshing mode of Fluent. You can display the polyhedral mesh, perform certain mesh manipulation operations, check the mesh quality, and so on.

> **Important:**
>
> You cannot read meshes from solvers that have been adapted using hanging nodes. To read one of these meshes in the meshing mode in Fluent, coarsen the mesh within the solver until you have recovered the original unadapted grid.

> **Note:**
>
> The naming of face zones can be controlled by Named Selections defined in Ansys Workbench. For details on exporting faceted geometry from Ansys Workbench, refer to the Ansys Workbench Help.

file/read-meshes-by-tmerge

Uses the **tmerge** utility to read the mesh contained in two or more separate files. It enables you to read the mesh files together and helps assemble the complete mesh.

file/read-multi-bound-mesh

Enables you to read multiple boundary mesh files into the meshing mode.

file/read-multiple-mesh

Enables you to read in two or more files together and have the complete mesh assembled for you, if the mesh files are contained in two or more separate files.

For example, if you are going to create a hybrid mesh by reading in a triangular boundary mesh and a volume mesh consisting of hexahedral cells, you can read both files at the same time using this command.

file/read-options

Enables you to set the following options for reading mesh files:

- Enforce mesh topology: This option is disabled by default. Enabling this option will orient the face zones consistently when the mesh file is read. If necessary, the zones being read will be separated, such that each boundary face zone has at most two cell zones as neighbors, one on either side. Also, internal face zones are inserted between neighboring cell zones that are connected by interior faces.

- Check read data: This option enables additional checks for the validity of the mesh. Enabling this option will check the mesh topology during file read. In case incorrect mesh topology is encountered, warning messages will be displayed and the erroneous entities will be deleted. Note that in case of mesh topology errors, no automatic mesh repair is done, and that parts of the mesh may be non-conformal, contain voids, or be erroneous in other ways. The purpose of the `check-read-data` option is to enable access to corrupt files. This option is disabled by default with the assumption that correct data will be read, and to shorten file read times.

`file/read-size-field`

Enables you to read in a size field file.

> **Note:**
>
> If you read a size-field file after scaling the model, ensure that the size-field file is appropriate for the scaled model (size-field vertices should match the scaled model).

`file/set-idle-timeout`

Allows you to set an idle timeout so that an idle Ansys Fluent session will automatically save and close after the specified time.

`file/set-tui-version`

Allows you to improve backwards compatibility for journal files. This command hides any new TUI prompts that are added at a future release of Ansys Fluent and reverts to the arguments of the release that you specify using the command (within two full releases of the current release). The command is automatically added to a journal file as soon as you start the recording. See Creating and Reading Journal Files for details.

`file/show-configuration`

Displays the current release and version information.

`file/start-journal`

Starts recording all input and writes it to a file. The current Fluent version is automatically recorded in the journal file. Note that commands entered using paths from older versions of Fluent will be upgraded to their current path in the journal file. See Creating and Reading Journal Files in the *Fluent User's Guide*.

To start the journaling process use the `file/start-journal` command, and end it with the `file/stop-journal` command (or by exiting the program). To read a journal file into the program, use the `file/read-journal` command.

> **Note:**
>
> The `read-journal` command always loads the file in the main (that is, top-level) menu, regardless of where you are in the menu hierarchy when you invoke it.

The standard period (.) alias is the same as the `file/read-journal` definition and is defined by:

```
(alias '. (lambda () (ti-read-journal)))
```

`file/start-transcript`

Starts recording input and output in a file.

A transcript file contains a complete record of all standard input to and output from Fluent (usually all keyboard and user interface input and all screen output).Start the transcription process with the `file/start-transcript` command, and end it with the `file/stop- transcript` command (or by exiting the program).

`file/`stop-journal

Stops recording input and closes the journal file.

`file/`stop-transcript

Stops recording input and output, and closes the transcript file.

`file/`write-boundaries

Enables you to write the specified boundaries into a mesh file.

This is useful for large cases where you may want to mesh different parts of the mesh separately and then merge them together. This enables you to avoid frequent switching between domains for such cases. You can write out selected boundaries to a mesh file and then create the volume mesh for the part in a separate session. You can then read the saved mesh into the previous session and merge the part with the rest of the mesh.

`file/`write-case

Enables you to write a case file that can be read by Fluent.

> **Note:**
>
> You should delete dead zones in the mesh before writing the mesh or case file for Fluent.

`file/`write-domains

Enables you to write all the mesh domains (except global) into a file that can be read.

`file/`write-mesh

Enables you to write a mesh file.

> **Note:**
>
> You should delete dead zones in the mesh before writing the mesh or case file for Fluent.

`file/`write-options

Enables you to set the enforce mesh topology option for writing mesh/case files.

This option is disabled by default; enabling this option will orient the face zones consistently when the mesh file is written. If necessary, the zones will be separated, such that each boundary face zone has at most two cell zones as neighbors, one on either side. Also, internal face zones will be inserted between neighboring cell zones that are connected by interior faces.

`file`/write-size-field

Enables you to write a size field file.

Chapter 7: `material-point/`

`material-point/`**create-material-point**

    Enables the definition of a material point. Specify the fluid zone name and the location to define the material point.

`material-point/`**delete-all-material-points**

    Enables the deletion of all defined material points.

`material-point/`**delete-material-point**

    Deletes the specified material point.

`material-point/`**list-material-points**

    Lists all the defined material points.

Chapter 8: `mesh/`

`mesh/`**auto-mesh**

Enables you to generate the volume mesh automatically. Specify a mesh object name for object-based auto mesh; if no name is given, face zone based auto mesh is performed. Specify the mesh elements to be used when prompted. Specify whether to merge the cells into a single zone or keep the cell zones separate. For face zone based meshing, specify whether automatically identify the domain to be meshed based on the topology information.

> **Note:**
>
> - You can specify the meshing parameters for the mesh elements (prisms, pyramids or non-conformals, tet, hex or poly) using either the respective dialog boxes or the associated text commands prior to using the `auto-mesh` command.

`mesh/`**auto-mesh-controls/**

Enters the auto-mesh-controls submenu

`mesh/auto-mesh-controls/`**backup-object**

Enables creation of a backup of the surface mesh before volume meshing starts. This option is enabled by default.

`mesh/`**auto-prefix-cell-zones**

Enables you to specify a prefix for cell zones created during the auto mesh procedure.

> **Note:**
>
> The `auto-prefix-cell-zones` command is not relevant for object-based meshing, where the cell zone names are generated based on the material points and the objects used to generate the mesh object.

`mesh/`**cavity/**

Enters the cavity menu.

`mesh/cavity/`**add-zones**

Enables you to create a cavity for adding new zones to the existing volume mesh.

mesh/cavity/create-hexcore-cavity-by-region

Creates the cavity in the hexcore mesh based on the zones and bounding box extents specified.

> **Note:**
>
> The `create-hexcore-cavity-by-region` option is no longer supported and will be removed at a future release.

mesh/cavity/create-hexcore-cavity-by-scale

Creates the cavity in the hexcore mesh based on the zones and scale specified.

> **Note:**
>
> The `create-hexcore-cavity-by-scale` option is no longer supported and will be removed at a future release.

mesh/cavity/merge-cavity

Enables you to merge the specified cavity domain with the parent domain.

During the merging operation, the cavity cell zones merges with the zones in the parent domain. The wall boundaries extracted from the interior zones will be converted to **interior** type and merged with the corresponding zones in the parent domain.

mesh/cavity/region

Enables you to create a cavity to modify the existing volume mesh in the specified region.

mesh/cavity/remove-zones

Enables you to create a cavity for removing zones from the existing volume mesh.

mesh/cavity/replace-zones

Enables you to create a cavity for removing a set of zones from an existing volume mesh and replacing them with new set of zones.

mesh/cell-zone-conditions/

Contains options for copying or clearing cell zone conditions when a case file is read.

mesh/cell-zone-conditions/clear

Clears the cell zone conditions assigned to the specified zones.

mesh/cell-zone-conditions/clear-all

Clears the cell conditions assigned to all the zones.

`mesh/cell-zone-conditions/copy`

Enables you to copy the cell zone conditions from the zone selected to the zones specified.

`mesh/check-mesh`

Checks the mesh for topological errors.

`mesh/check-quality`

Enables you to ensure that the mesh quality is appropriate before transferring the mesh to the solution mode.

`mesh/check-quality-level`

Enables you to report additional quality metrics when set to 1.

In addition to the orthogonal quality and Fluent aspect ratio, additional metrics such as cell squish and skewness will be reported when the `check-quality-level` is set to **1**.

`mesh/clear-mesh`

Enables you to generate a new mesh by deleting the internal mesh and leaving only the boundary faces and nodes.

`mesh/create-heat-exchanger`

Creates the heat exchanger mesh. You need to specify the method for selecting the Location coordinates (by Position or Nodes), the location coordinates, the parameters for setting up mesh density (by Interval or Size), and the number of intervals (sizes) between points (nodes) 1–2, 1–3, 1–4. Also specify the object/zone name prefix and enable creating the mesh object, if required.

`mesh/cutcell/`

Enters the cutcell menu.

> **Note:**
>
> This menu is no longer supported, and will be removed in a future release.

`mesh/cutcell/create`

Creates the CutCell mesh by performing the initialize, refine, snap, and improve operations sequentially.

`mesh/cutcell/create-prism`

Creates the prism layers on the recovered boundary based on the zone-specific prism parameters set. Specify the cell zones into which the prism layers are to be grown and the gap factor as appropriate.

`mesh/cutcell/modify/`

Enters the cutcell modify menu.

`mesh/cutcell/modify/auto-node-move`

Enables you to use the **Auto Node Move** utility to improve the CutCell mesh quality.

`mesh/cutcell/modify/cavity-remeshing`

Enables you to use the **Cavity Remeshing** utility to improve the CutCell mesh quality near the boundary.

> **Note:**
>
> Face zones of type **internal** are recovered as type **wall** in the cutcell mesher. These should be reset to type **internal** before using the cavity remesher.

`mesh/cutcell/modify/post-morph-improve`

Improves the quality of the CutCell mesh post-prism generation.

`mesh/cutcell/modify/rezone-multi-connected-faces`

Enables you to resolve multi-connected configurations on the CutCell boundary. Specify an appropriate value for the critical count for contiguous manifold faces.

An example is shown in Figure 8.1: Rezoning Multiply Connected Faces (p. 108) where the multiply connected faces around the surface are removed.

Figure 8.1: Rezoning Multiply Connected Faces
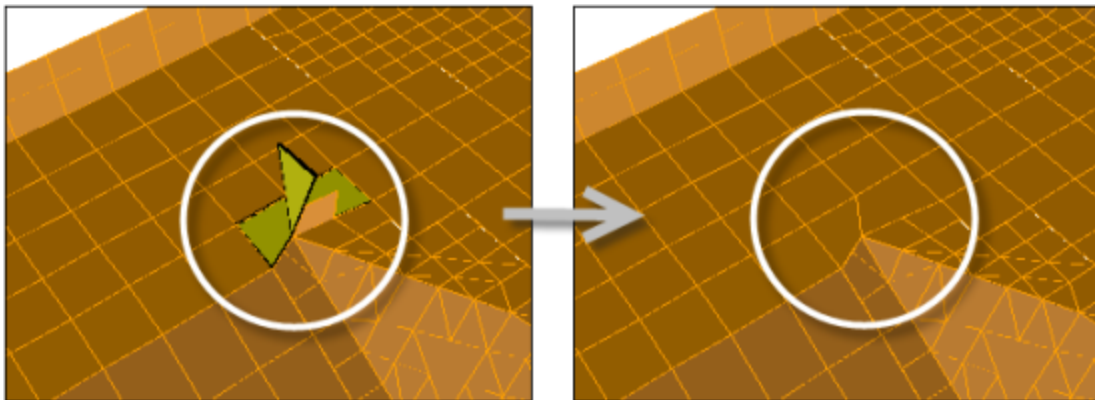
`mesh/cutcell/modify/split-boundary`

Creates a copy of the specified CutCell boundary zones and makes the boundary mesh conformal at the hanging-nodes on the copied zones. The new zones will be named based on the original zone names and prefixed by `split-`.

`mesh/cutcell/objects/`

Enters the `objects` menu.

mesh/cutcell/objects/change-object-type

Enables you to change the object type (`geom` or `mesh`).

mesh/cutcell/objects/check-mesh

Checks the mesh on the specified objects for connectivity and orientation of faces. The domain extents, volume statistics, and face area statistics will be reported along with the results of other checks on the mesh.

mesh/cutcell/objects/create

Creates the object based on the priority, cell zone type, face zones, edge zones, and object type specified. You can specify the object name or retain the default blank entry to have the object name generated automatically.

mesh/cutcell/objects/create-and-activate-domain

Creates and activates the domain comprising the face zones from the objects specified.

mesh/cutcell/objects/create-groups

Creates a face group and an edge group comprising the face zones and edge zones included in the specified objects, respectively.

mesh/cutcell/objects/create-intersection-loops

Enables you to create intersection loops for objects.

- The `collectively` option creates an interior edge loop at the intersection between two adjacent face zones included in the same object and between multiple objects.

- The `individually` option creates an interior edge loop at the intersection between two adjacent face zones included in the same object.

mesh/cutcell/objects/create-multiple

Creates multiple objects by creating an object per face zone specified. The objects will be named automatically based on the prefix and priority specified.

mesh/cutcell/objects/create-new-mesh-object/

Contains options for creating a new mesh object by wrapping or remeshing existing objects.

mesh/cutcell/objects/create-new-mesh-object/remesh

Creates a new mesh object by remeshing geometry objects individually or collectively.

mesh/cutcell/objects/create-new-mesh-object/wrap

Creates a new mesh object by wrapping the specified objects individually or collectively.

mesh/cutcell/objects/delete

Deletes the specified objects.

`mesh/cutcell/objects/`delete-all

Deletes all the defined objects.

`mesh/cutcell/objects/`delete-all-geom

Deletes all the defined `geom` objects.

`mesh/cutcell/objects/`delete-unreferenced-faces-and-edges

Deletes all the faces and edges that are not included in any defined objects.

`mesh/cutcell/objects/`extract-edges

Extracts the edge zones from the face zones included in the specified objects, based on the edge-feature-angle value specified (`/mesh/cutcell/objects/set/set-edge-feature-angle`).

`mesh/cutcell/objects/`improve-feature-capture

Enables you to imprint the edges comprising the object on to the object face zones to improve feature capture for mesh objects. You can specify the number of imprinting iterations and additional aggressive imprinting iterations to be performed.

`mesh/cutcell/objects/`improve-object-quality

This command is not relevant for CutCell meshing.

`mesh/cutcell/objects/`join-intersect/

The commands in this sub-menu are not relevant for CutCell meshing.

`mesh/cutcell/objects/`list

Lists the defined objects, indicating the respective cell zone type, priority, face zones and edge zones comprising the object, object type, and object reference point in the console.

`mesh/cutcell/objects/`merge

Merges the specified objects into a single object.

`mesh/cutcell/objects/`merge-edges

Merges all the edge zones in an object into a single edge zone.

> **Note:**
>
> If the object is composed of edge zones of different types (boundary and interior), the edge zones of the same type (boundary or interior) will be merged into a single edge zone.

mesh/cutcell/objects/merge-nodes

Merges the free nodes at the object level based on the specified tolerance or using a tolerance that is a specified percentage of shortest connected edge length.

mesh/cutcell/objects/merge-voids

Enables you to merge voids in the mesh object after the sewing operation. This command is not relevant for CutCell meshing.

mesh/cutcell/objects/merge-walls

Merges all the face zones of type `wall` in an object into a single face zone.

mesh/cutcell/objects/remove-gaps/

Contains options for removing gaps between mesh objects. The commands in this sub-menu are not relevant for CutCell meshing.

mesh/cutcell/objects/rotate

Rotates the objects based on the angle of rotation, pivot point, and axis of rotation specified.

mesh/cutcell/objects/scale

Scales the objects based on the scale factors specified.

mesh/cutcell/objects/separate-faces-by-angle

Separates the face zones comprising the object based on the angle specified.

mesh/cutcell/objects/separate-faces-by-seed

Separates the face zones comprising the object based on the seed face specified.

mesh/cutcell/objects/set/

Contains the following options:

mesh/cutcell/objects/set/set-edge-feature-angle

Sets the edge feature angle to be used for extracting edge zones from the face zones included in the objects.

mesh/cutcell/objects/set/show-edge-zones?

Displays the edge zones comprising the objects drawn in the graphics window.

mesh/cutcell/objects/set/show-face-zones?

Displays the face zones comprising the objects drawn in the graphics window.

`mesh/cutcell/objects/`sew/

Contains options for sewing mesh objects. The commands in this sub-menu are not relevant for CutCell meshing.

`mesh/cutcell/objects/`translate

Translates the objects based on the translation offsets specified.

`mesh/cutcell/objects/`update

Enables you to update the objects defined when the face and/or edge zones comprising the object have been deleted.

`mesh/cutcell/objects/`wrap/

Contains options for the object wrapping operation. The commands in this sub-menu are not relevant for CutCell meshing.

`mesh/cutcell/`set/

Enters the CutCell settings menu.

`mesh/cutcell/set/`auto-delete-dead-zones?

Controls the automatic deleting of the dead zones in the CutCell mesh.

`mesh/cutcell/set/`auto-delete-solid-zones?

Controls the automatic deleting of the solid zones in the CutCell mesh.

`mesh/cutcell/set/`create-material-point

Enables you to define a material point.

`mesh/cutcell/set/`delete-all-material-points

Enables you to delete all defined material points.

`mesh/cutcell/set/`delete-material-point

Deletes the specified material point.

`mesh/cutcell/set/`list-material-points

Lists all the defined material points.

`mesh/cutcell/set/`max-initial-cells

Specifies the maximum number of cells in the initial Cartesian grid.

`mesh/cutcell/set/`set-cutcell-quality-method

Enables you to set the quality measure for the improve operation. The default measure used is the orthoskew metric.

mesh/cutcell/set/set-post-morph-parameters

Enables you to set parameters for improving the CutCell mesh post-prism generation using the command `/mesh/cutcell/modify/post-morph-improve`.

mesh/cutcell/set/set-post-snap-parameters

Enables you to set parameters for improving the CutCell mesh quality.

mesh/cutcell/set/set-thin-cut-face-zones

Enables you to specify the face zones constituting the thin regions to be recovered during the CutCell meshing process.

mesh/cutcell/set/set-thin-cut-edge-zones

Enables you to specify the edge zones defining the features in thin regions to be recovered during the CutCell meshing process.

mesh/cutcell/size-functions/

Enters the size functions menu.

mesh/cutcell/size-functions/compute

Computes the size field based on the defined parameters.

mesh/cutcell/size-functions/contours/

Enters the contours sub-menu.

mesh/cutcell/size-functions/contours/draw

Displays contours in the graphics window. Compute the size field using `/size-functions/compute` or read in a size field file prior to displaying the contours of size.

mesh/cutcell/size-functions/contours/set/refine-facets?

Enables you to specify smaller facets if the original are too large. Default is `no`.

mesh/cutcell/size-functions/create

Defines the size function based on the specified parameters.

mesh/cutcell/size-functions/create-defaults

Creates default size functions based on face and edge curvature and proximity.

mesh/cutcell/size-functions/delete

Deletes the specified size function or the current size field.

mesh/cutcell/size-functions/delete-all

Deletes all the defined size functions.

mesh/cutcell/size-functions/list

Lists all the defined size functions and the corresponding parameter values defined.

mesh/cutcell/size-functions/list-periodicity-filter

Lists the details of the source zone and rotational periodic parameters specified for the size field.

mesh/cutcell/size-functions/reset-global-controls

Resets the values for the global controls to the defaults.

mesh/cutcell/size-functions/set-global-controls

Sets the values for the global minimum and maximum size, and growth rate.

mesh/cutcell/size-functions/set-periodicity-filter

Enables you to apply periodicity to the size field by selecting one source face zone.

> **Note:**
>
> Ensure that periodicity is previously defined in the **Make Periodic Boundaries** dialog box.
>
> Only rotational periodicity is supported, translational periodicity is not supported currently.

mesh/cutcell/size-functions/set-prox-gap-tolerance

Sets the tolerance relative to minimum size to take gaps into account. Gaps whose thickness is less than the global minimum size multiplied by this factor will not be regarded as a proximity gap.

mesh/cutcell/size-functions/set-scaling-filter

Enables you specify the scale factor, and minimum and maximum size values to filter the size output from the size field.

mesh/cutcell/size-functions/triangulate-quad-faces?

Identifies the zones comprising non-triangular elements and uses a triangulated copy of these zones for computing the size functions.

mesh/cutcell/size-functions/un-set-periodicity-filter

Removes periodicity from the size field.

mesh/cutcell/size-functions/use-cad-imported-curvature?

Allows you to use curvature data from the nodes of the CAD facets.

`mesh/domains/`

Enters the domain menu.

`mesh/domains/`activate

Activates the specified domain for meshing or reporting operations.

`mesh/domains/`create

Creates a new domain based on the specified boundary face zones. Ensure valid boundary zones are specified; specifying invalid zones will generate an error.

`mesh/domains/`create-by-cell-zone

Creates a new domain based on the specified cell zone.

`mesh/domains/`create-by-point

Creates a new domain based on the specified

> **Note:**
>
> The create-by-point option works only for cases with no overlapping face zones.

`mesh/domains/`delete

Deletes the specified domain.

`mesh/domains/`draw

Displays the boundary face zones of the specified domain.

`mesh/domains/`print

Prints the information for the specified domain.

`mesh/hexcore/`

Enters the hexcore menu.

`mesh/hexcore/`controls/

Enters the hexcore controls menu.

`mesh/hexcore/controls/`avoid-1:8-cell-jump-in-hexcore

Determines the types of cells (large cells, small cells, or both) that are going to be replaced with pyramid and tet cells in order to avoid any 1:8 cell jumps in the hexcore region. The default is `no` and `small-cells`.

`mesh/hexcore/controls/`buffer-layers

Sets the number of addition cells to mark for subdivision.

`mesh/hexcore/controls/`compute-max-cell-length

Computes the maximum cell length for the hexcore mesh.

`mesh/hexcore/controls/`define-hexcore-extents?

Enables you to extend the hexcore mesh to specified domain extents and/or selected planar boundaries. When enabled, the `outer-domain-params` sub-menu will be available.

`mesh/hexcore/controls/`delete-dead-zones?

Toggles the automatic deleting of the dead zones.

`mesh/hexcore/controls/`maximum-cell-length

Sets the maximum cell length for the hex cells in the domain.

`mesh/hexcore/controls/`maximum-initial-cells

Specifies the maximum number of cells in the initial Cartesian mesh.

`mesh/hexcore/controls/`non-fluid-type

Selects the default non-fluid cell zone type. After the mesh is initialized, any non-fluid zones will be set to this type. If the mesh includes multiple regions (for example, the problem for which you are creating the mesh includes a fluid zone and one or more solid zones), and you plan to refine all of them using the same refinement parameters, modify the **Non-Fluid Type** *before* generating the hexcore mesh.

Note:

For zone-based meshing, if any cell zone has at least one boundary zone type as inlet, it will automatically be set to fluid type. For object based meshing, volume region type is used to determine the cell zone type.

`mesh/hexcore/controls/`octree-hexcore?

Speeds up hexahedral core generation by enabling the octree technique for hexcore mesh generation. This option is disabled by default.

Body-of-influence sizing may be used for refinement.

This option does not support hexcore generation up to boundaries.

`mesh/hexcore/controls/`outer-domain-params/

Contains options for defining the outer domain parameters. This sub-menu is available only when `define-hexcore-extents?` is enabled.

`mesh/hexcore/controls/outer-domain-params/`specify-coordinates?

Enables you to specify the extents of the hexcore outer box using the `coordinates` command.

---

mesh/hexcore/controls/outer-domain-params/coordinates

Specifies the extents (min and max coordinates) of the hexcore outer box. This command is available when the `specify-coordinates?` option is enabled.

mesh/hexcore/controls/outer-domain-params/specify-boundaries?

Enables you to specify selected boundaries to which the hexcore mesh is to be generated using the `boundaries` command.

mesh/hexcore/controls/outer-domain-params/boundaries

Specifies the boundaries to which the hexcore mesh is to be generated when the `specify-boundaries?` option is enabled. After specifying the boundaries, the `auto-align?`, `delete-old-face-zones?`, and `list` options will also be available.

mesh/hexcore/controls/outer-domain-params/auto-align?

Enables you to axis-align non-aligned planar boundaries to which hexcore mesh is to be generated. This option is available only when the `specify-boundaries?` option is enabled and the boundaries are specified.

mesh/hexcore/controls/outer-domain-params/auto-align-tolerance

Specifies the tolerance for aligning boundary zones when `auto-align?` is enabled.

mesh/hexcore/controls/outer-domain-params/auto-align-boundaries

Aligns the boundary zones specified (using the `boundaries` command) with the tolerance specified \ (using the `auto-align-tolerance` command) when `auto-align?` is enabled.

mesh/hexcore/controls/outer-domain-params/delete-old-face-zones?

Enables you to delete the original tri face zones that have been replaced during the hexcore meshing process. This option is available only when the `specify-boundaries?` option is enabled and the boundaries are specified.

mesh/hexcore/controls/outer-domain-params/list

Lists the boundaries to which the hexcore mesh is to be generated. This option is available only when the `specify-boundaries?` option is enabled and the boundaries are specified.

mesh/hexcore/controls/peel-layers

Specifies the distance for the hexcore interface to peel-back from the boundary. The default value is 0. The higher the value of peel layer, the bigger the distance between the hexcore interface and the boundary.

mesh/hexcore/controls/print-region-based-sizing

Displays local sizing settings (max cell length and growth rate) for specified region(s).

mesh/hexcore/controls/set-region-based-sizing

Allows you to specify local sizing settings (max cell length and growth rate) for specified region(s).

mesh/hexcore/controls/skip-tet-refinement?

Enables you to omit the tetrahedral refinement phase for reducing total cell count (default is no). Hex cell count is unaffected.

Tip:

You should use `peel-layers` equal 0 when this option is enabled.

> **Warning:**
>
> This option may leave narrow regions without hexcore mesh. Tetrahedral cells with lower quality may be created in such regions, reducing the overall mesh quality.

mesh/hexcore/create

Enables you to create the hexcore mesh according to the specified parameters.

mesh/hexcore/local-regions/

Enters the hexcore local refinement region sub-menu.

mesh/hexcore/local-regions/activate

Enables you to activate the specified local regions for refinement.

mesh/hexcore/local-regions/deactivate

Enables you to deactivate the specified local regions for refinement.

mesh/hexcore/local-regions/define

Defines the local region according to the specified parameters.

mesh/hexcore/local-regions/delete

Deletes the specified refinement region.

mesh/hexcore/local-regions/ideal-hex-vol

Reports the ideal hex volume for the given edge length.

mesh/hexcore/local-regions/init

Creates a default region encompassing the entire geometry.

mesh/hexcore/local-regions/list-all-regions

Lists the defined and active regions in the console.

mesh/hexcore/merge-tets-to-pyramids?

Enables the `merge-tets-to-pyramids` command to reduce the total cell count.

If `skip-tet-refinement` is enabled, pairs of tets will be merged into pyramids. Hexcore count is unaffected.

mesh/laplace-smooth-nodes

Applies a Laplacian smoothing operator to the mesh nodes. This command can be used for smoothing of all cell types, including prismatic cells.

mesh/list-mesh-parameter

Shows all mesh parameters.

mesh/manage/

Enters the manage cell zones menu.

mesh/manage/active-list

Lists all active zones.

mesh/manage/adjacent-face-zones

Lists all face zones that refer to the specified cell zone.

mesh/manage/auto-set-active

Sets the active zones based on points that are defined in an external file. For each zone you want to activate, you need to specify the coordinates of a point in the zone, the zone type (for example, fluid), and (optionally) a new name. A sample file is shown below:

```
((1550.50 -466.58 896.41) fluid heater-#)
((1535.83 -643.14 874.71) fluid below-heater-#)
((1538.73 -444.28 952.69) fluid above-heater-#)
((1389.18 -775.51 825.97) fluid plenum-#)
```

Here, four fluid zones are identified, renamed, and activated. Any zone that you identify in the file will automatically be activated. The # indicates that the mesher should append the appropriate ID number for the zone.

> **Warning:**
>
> This command is valid only for tet meshes.

mesh/manage/change-prefix

Enables you to change the prefix for the cell zone.

mesh/manage/copy

Copies all nodes and faces of specified cell zones.

mesh/manage/delete

Deletes a cell zone, along with its associated nodes and faces. When deleting cell zones that contain poly cells, you will be warned that the original mesh needs to be deleted and the original faces restored prior to remeshing the volumetric region.

mesh/manage/get-material-point

Prints the coordinates of the material point for the specified cell zone.

> **Note:**
>
> If the cell zone is non-contiguous, the `get-material-point` command will print a list of material points, one for each contiguous region.

mesh/manage/id

Specifies a new cell zone ID. If a conflict is detected, the change will be ignored.

mesh/manage/list

Prints information on all cell zones.

mesh/manage/merge

Merges two or more cell zones.

> **Note:**
>
> For object-based merge, the selected zones must be in the same volumetric region. If not, you will have to merge the volumetric regions first using `/objects/volumetric-regions/merge`. If the volumetric regions cannot be merged because they are not contiguous, you will have to delete the object(s) only before merging the cell zones.

mesh/manage/merge-dead-zones

Enables you to merge dead zones having a cell count lower than the specified threshold value, with the adjacent cell zone. The result of the merge operation is determined by the type of the adjacent cell zone and the shared face area. The priority for merging with the adjacent cell zone based on type is fluid > solid > dead (that is, merging with an adjacent fluid zone takes priority over merging with an adjacent solid zone, which in turn takes priority over merging with a dead zone). Also, if the adjacent zones are of the same type (for example, fluid), the zone will be merged with the zone having the largest shared face area.

mesh/manage/name

Enables you to rename a cell zone.

`mesh/manage/`origin

Specifies a new origin for the mesh, to be used for cell zone rotation. The default origin is (0,0,0).

`mesh/manage/`revolve-face-zone

Generates cells by revolving a face thread.

`mesh/manage/`rotate

Rotates all nodes of specified cell zones by a specified angle.

`mesh/manage/`rotate-model

Rotates all nodes of the model by a specified angle.

`mesh/manage/`scale

Scales all nodes of specified cell zones by a specified factor.

`mesh/manage/`scale-model

Scales all nodes of the model by a specified factor.

`mesh/manage/`set-active

Sets the specified cell zones to be active.

`mesh/manage/`translate

Translates all nodes of specified cell zones by a specified vector.

`mesh/manage/`translate-model

Translates all nodes of the model by a specified vector.

`mesh/manage/`type

Changes the type and name of a cell zone.

`mesh/`modify/

Enters the mesh modify menu.

`mesh/modify/`auto-improve-warp

Enables you to improve face warp by node movement. Specify the appropriate cell zones and boundary zones, the maximum warp, the number of iterations per face to be improved, and the number of iterations of the automatic node movement procedure (default, 4).

`mesh/modify/`auto-node-move

Enables you to improve the mesh quality by node movement. Specify the appropriate cell zones and boundary zones, the quality limit based on the quality measure selected, dihedral angle, the number of iterations per node to be moved and the number of iterations of the automatic

node movement procedure (default, 1). You can also choose to restrict the movement of boundary nodes along the surface.

mesh/modify/clear-selections

Clears all items from the selection list.

mesh/modify/deselect-last

Deselects the last item you selected using the `select-entity` command.

mesh/modify/extract-unused-nodes

Places all unused nodes in a separate interior node zone.

mesh/modify/list-selections

Lists all items in the selection list.

mesh/modify/list-skewed-cells

Lists cells with skewness in a specified range.

mesh/modify/mesh-node

Attempts to introduce a new node into the existing mesh.

mesh/modify/mesh-nodes-on-zone

Inserts nodes associated with node or face zone into the volume mesh.

> **Important:**
>
> If a face zone is specified, the faces are deleted before the nodes are introduced into the mesh.

mesh/modify/neighborhood-skew

Reports the maximum skewness of cells using the specified node.

mesh/modify/refine-cell

Attempts to refine the cells in the probe list by introducing a node nears its centroid. This technique is useful for removing very flat cells near the boundary when boundary sliver removal is not possible. After refining the cell, you should smooth the mesh.

mesh/modify/repair-negative-volume-cells

Repairs negative volume cells by moving nodes. Specify the appropriate boundary zones, the number of iterations per node to be moved, dihedral angle, whether to restrict the movement of boundary nodes along the surface, and the number of iterations of the automatic node movement procedure (default, 1).

`mesh/modify/select-entity`

Adds an entity (face, node, cell, etc.) to the selection list.

`mesh/modify/smooth-node`

Applies Laplace smoothing to the nodes in the selection list.

`mesh/non-conformals/`

Enters the non-conformals menu.

`mesh/non-conformals/controls/`

Enters the non-conformals controls menu.

`mesh/non-conformals/controls/enable?`

Toggles the creation of a non-conformal interface.

`mesh/non-conformals/controls/retri-method/`

Specifies the method to be used for retriangulating the quad faces on the non-conformal zones.

`mesh/non-conformals/controls/retri-method/prism`

Remeshes the prism-side quad zones named **prism-side*** or ***-quad***.

`mesh/non-conformals/controls/retri-method/quad-split`

Splits the quad faces diagonally into tri faces.

`mesh/non-conformals/controls/retri-method/remesh`

Remeshes all the quad faces based on the edge and surface feature angle specified.

`mesh/non-conformals/create`

Creates the non-conformal interface on the specified face zones using the specified retriangulation method.

`mesh/non-conformals/separate`

Enables you to separate the face zones comprising the non-conformal interface between the cell zones specified. Specify the cell zones where the interface is non-conformal, an appropriate gap distance, and the critical angle to be used for separating the face zones. You can also choose

to orient the boundary face zones after separation and additionally write a journal file for the separation operation.

> **Note:**
>
> If you choose to write a journal file when using the `/mesh/non-conformals/sep-arate` command to separate the mesh interface zones, you can read the journal file to create the mesh interface automatically in solution mode.

mesh/**poly/**

Enters the polyhedral mesh generation menu.

mesh/poly/**collapse**

Merge nodes to remove short edges and small faces. The decision threshold uses **edge size ratio**, **face size ratio**, and (face) **area fraction.**

mesh/poly/**controls/**

Enters the controls menu for setting poly parameters.

mesh/poly/controls/**cell-sizing**

Sets cell volume distribution function as `geometric`, `linear`, or `size-field`.

mesh/poly/controls/**edge-size-ratio**

Sets the threshold for the size ratio of two connected edges. Recommended range is 20 to 200.

mesh/poly/controls/**face-size-ratio**

Sets the threshold for the size ratio of two faces on one cell. Recommended range is 100 to 300.

mesh/poly/controls/**feature-angle**

Sets the minimum threshold that should be preserved as a feature.

mesh/poly/controls/**improve?**

Enables poly mesh improvement by smoothing based on the `smooth-controls`.

mesh/poly/controls/**merge-skew**

Sets the minimum skewness threshold for cell merge.

mesh/poly/controls/**non-fluid-type**

Selects the default type for non-fluid zones.

mesh/poly/controls/remesh-skew

Sets the target skewness when remeshing.

mesh/poly/controls/sliver-cell-area-fraction

Sets the threshold for the area of a single face to the cell surface area. Recommended range is 0.00001 to 0.001.

mesh/poly/controls/smooth-controls/

Enters the menu for setting smoothing parameters for poly mesh.

mesh/poly/controls/smooth-controls/centroid-smooth-iterations

Sets the number of passes for tet-cell centroid smoothing during the poly mesh generation phase.

mesh/poly/controls/smooth-controls/edge-smooth-iterations

Sets the number of passes for tet-cell edge smoothing during the poly mesh generation phase.

mesh/poly/controls/smooth-controls/laplace-smooth-iterations

Sets the number of passes for tet-cell Laplace smoothing during the poly mesh generation phase.

mesh/poly/controls/smooth-controls/smooth-attempts

Sets the maximum number of movements for a single node during poly mesh smoothing.

mesh/poly/controls/smooth-controls/smooth-boundary

Enables boundary smoothing as part of poly cell smoothing. Default is no.

mesh/poly/controls/smooth-controls/smooth-iterations

Sets the number of improvement passes over the full poly mesh.

mesh/poly/controls/smooth-controls/smooth-on-layer

Constrains movement of nodes to maintain layering during poly mesh smoothing.

mesh/poly/controls/smooth-controls/smooth-skew

Sets the minimum skewness threshold for poly mesh smoothing.

mesh/poly/improve

Allows you to improve the polyhedral mesh quality based on the quality-method.

mesh/poly/local-regions/

Enters the local refinement menu.

> **Note:**
>
> Poly meshing follows tet meshing. These commands behave like the equivalent commands under `/mesh/tet/local-regions/`.

mesh/poly/local-regions/activate

Activates the specified regions for refinement.

mesh/poly/local-regions/deactivate

Deactivates the specified regions for refinement.

mesh/poly/local-regions/define

Defines the refinement region according to the specified parameters.

mesh/poly/local-regions/delete

Deletes the specified refinement region.

mesh/poly/local-regions/ideal-vol

Reports the volume of an ideal tetrahedron for the edge length specified.

mesh/poly/local-regions/init

Defines the default refinement region encompassing the entire geometry.

mesh/poly/local-regions/list-all-regions

Lists all refinement region parameters and the activated regions in the console.

mesh/poly/local-regions/refine

Refines the active cells inside the selected region based on the specified refinement parameters.

mesh/poly/quality-method

Asks you to choose from `internal-default`, `orthoskew` or `squish` quality measure for mesh improvement.

mesh/poly/remesh

Improves the quality in a local region based on the minimum skewness threshold.

mesh/poly-hexcore/

Enters the menu for poly-hexcore mesh.

mesh/poly-hexcore/controls/

Enters the menu for setting parameters for poly-hexcore mesh.

mesh/poly-hexcore/controls/avoid-1:8-cell-jump-in-hexcore

Determines the types of cells (large cells, small cells, or both) that are going to be replaced with polyhedra cells in order to avoid any 1:8 cell jumps in the hexcore region. The default is `no` and `large-cells`.

mesh/poly-hexcore/controls/mark-core-region-cell-type-as-hex?

Determines whether or not to apply hexahedra cells in the core region of the mesh. The default value is `yes`.

mesh/poly-hexcore/controls/only-polyhedra-for-selected-regions

Determines if polyhedra cells are to be applied to the selected regions.

mesh/prepare-for-solve

Prepares the mesh for solving in solution mode by performing a cleanup operation after the volume mesh has been generated. Operations such as deleting dead zones, deleting geometry objects, deleting edge zones, deleting unused faces and nodes are performed during this operation.

mesh/prism/

Enters the prism menu.

mesh/prism/controls/

Enters the prism controls menu.

mesh/prism/controls/adjacent-zone/

Enters the prism adjacent zone controls menu.

mesh/prism/controls/adjacent-zone/project-adjacent-angle

Determines whether or not to project to an adjacent zone. If a zone shares outer nodes with any of the zones from which the layers are being grown (the "base zones"), its angle with respect to the growth direction is compared with this value. If the angle is less than or equal to this value, then the zone will be projected to. The default value is 75 degrees. See Using Adjacent Zones as the Sides of Prisms for details.

mesh/prism/controls/adjacent-zone/side-feature-angle

Specifies the angle used for computing the feature normals.

mesh/prism/controls/check-quality?

Enables/disables the checking of volume, skewness, and handedness of each new cell and face.

`mesh/prism/controls/`improve/

Enters the prism smoothing controls menu.

`mesh/prism/controls/improve/`check-allowable-skew?

Enables you to check the skewness of the prism cap for every layer.

`mesh/prism/controls/improve/`corner-height-weight?

When enabled, the offset height at corners with large angles (for example, 270º) is reduced to give a smoother prism cap.

`mesh/prism/controls/improve/`edge-swap-base-angle

Specifies the maximum allowable angle between the normals of the base faces for skewness-driven edge swapping.

`mesh/prism/controls/improve/`edge-swap-cap-angle

Specifies the maximum allowable angle between the normals of the cap faces for skewness-driven edge swapping.

`mesh/prism/controls/improve/`improve-warp?

Enables or disables improving of face warp during prism generation. This option is disabled by default.

`mesh/prism/controls/improve/`left-hand-check

Controls checking for left-handedness of faces. The default setting of `0` implies face handedness will not be checked. A value of `1` implies only cap faces will be checked, while `2` implies faces of all cells in current layer will be checked.

`mesh/prism/controls/improve/`max-allowable-cap-skew

Specifies the maximum skewness allowed for a prism cap face. If the skewness of a cap face exceeds this value, the meshing process will stop and a warning indicates that the skewness for that layer is too high.

`mesh/prism/controls/improve/`max-allowable-cell-skew

Specifies the cell quality criteria for smoothing and quality checking.

`mesh/prism/controls/improve/`smooth-improve-prism-cells?

Enables you to set the parameters for improving the prism cells after the required prism layers are created. You can select optimized smoothing (`smooth`), node movement (`improve`), or a combination of both to improve the quality. Specify the quality measure to be used, the cell quality threshold, the number of improvement iterations, and the minimum improvement required.

`mesh/prism/controls/`merge-ignored-threads?

Enables you to automatically merge all ignored zones related to a base thread into a single thread. This option is enabled by default. When this option is disabled, more than one ignored thread will be generated per base thread. However, various zones can be created by ignoring this option. They are:

`mesh/prism/controls/`merge-ignored-threads?`*`:dwall

Is formed due to dangling wall.

`mesh/prism/controls/`merge-ignored-threads?`*`:prox

Is formed due to proximity.

`mesh/prism/controls/`merge-ignored-threads?`*`:ud_normal

Is formed due to invalid normal.

`mesh/prism/controls/`merge-ignored-threads?`*`:smooth

Is formed due to smoothing of ignored regions.

The default value of the tgvar `prism/ignore-extension` (for which there is no TUI command) is 1. It means that after the prism generation is over, all the ignored regions are expanded by this number. If we expand say by n times, that is if `prism/ignore-extension` is set to n using `tgsetvar!` (with the exclamation mark), more ignored threads will be created because of expanding ignored regions. These regions will be named as follows:

`*:cell_delete_1, *:cell_delete_2, *:cell_delete_3,....` up to \
`*:cell_delete_n`

If `merge-ignored-threads?` is enabled, then all these are merged and you finally get only one zone per base thread, named as `*:ignore`.

`mesh/prism/controls/`morph/

Enters the prism morphing controls menu.

`mesh/prism/controls/morph/`improve-threshold

Specifies the quality threshold used for improving the quality during the morphing operation.

`mesh/prism/controls/morph/`morphing-convergence-limit

Specifies the convergence limit for the morphing operation. The morpher uses an iterative solver. It is assumed to have converged when the relative residual is less than this number.

`mesh/prism/controls/morph/`morphing-frequency

Specifies the frequency of the morphing operation. The number specified denotes the number of prism layers after which the morpher is applied to the remainder of the mesh

(for example, a value of 5 indicates that the morpher is applied to the mesh after every 5 prism layers grown).

mesh/prism/controls/normal/

Enters the prism normal controls menu.

mesh/prism/controls/normal/bisect-angle

Is required for growing prisms out of sharp interior corners. When the value of this angle is set, the normals are automatically projected onto the plane bisecting the angle between faces having an interior angle less than this angle.

mesh/prism/controls/normal/compute-normal

Computes the normal for the specified face zone.

mesh/prism/controls/normal/direction-method

Specifies whether the prism layers should be grown normal to surfaces or along a specified direction vector.

mesh/prism/controls/normal/direction-vector

Specifies the direction vector for prism extrusion when the `uniform` method is selected for `direction-method`.

mesh/prism/controls/normal/ignore-invalid-normals?

Enables you to ignore nodes that have poor normals.

mesh/prism/controls/normal/max-angle-change

Specifies the maximum angle by which the normal direction at a node can change during smoothing.

mesh/prism/controls/normal/orient-mesh-object-face-normals

Enables you to orient the face normals for mesh object boundary zones. Specify the mesh object, region or material point as appropriate, and specify whether walls, baffles or both comprising the prism base zones are to be separated and oriented.

mesh/prism/controls/normal/orthogonal-layers

Specifies the number of layers to preserve orthogonality. All smoothing is deferred until after these layers.

mesh/prism/controls/offset/

Enters the prism offset controls menu.

mesh/prism/controls/offset/first-aspect-ratio-min

Specifies the minimum first aspect ratio (ratio of prism base length to prism layer height) for the prism cells.

`mesh/prism/controls/offset/`min-aspect-ratio

Specifies the minimum aspect ratio (ratio of prism base length to prism layer height) for the prism cells.

`mesh/prism/controls/`post-ignore/

Contains options for setting the parameters for removing poor quality prism cells after the required prism layers are created.

`mesh/prism/controls/post-ignore/`post-remove-cells?

Enables you to set the parameters for removing poor quality prism cells after the required prism layers are created. You can remove cells based on quality, intersection, interior warp, and feature edges. Specify options for removing additional cells in regions of high aspect ratio and feature angle, the number of cell rings to be removed around the marked cells, and options for smoothing the prism boundary and prism side height.

`mesh/prism/controls/`proximity/

Enters the prism proximity controls menu.

`mesh/prism/controls/proximity/`allow-ignore?

Enables you to ignore nodes where the specified maximum shrink factor cannot be maintained.

`mesh/prism/controls/proximity/`allow-shrinkage?

Enables shrinkage while growing prism layers.

`mesh/prism/controls/proximity/`gap-factor

Controls the gap between the intersecting prisms layers in the proximity region with respect to the cell size of the prisms.

`mesh/prism/controls/proximity/`keep-first-layer-offsets?

Enables you to retain first layer offsets while performing proximity detection.

`mesh/prism/controls/proximity/`max-aspect-ratio

Specifies the maximum allowable cell aspect ratio to determine the limit for the shrinkage of prism layers. This option is available only when the `allow-ignore?` option is disabled.

`mesh/prism/controls/proximity/`max-shrink-factor

Specifies the shrink factor determining the maximum shrinkage of the prism layers. This option is available only when the `allow-ignore?` option is enabled.

`mesh/prism/controls/`remove-invalid-layer?

Removes the last prism layer if it fails in the quality check.

`mesh/prism/controls/`set-post-mesh-controls

Sets controls specific to growing prisms post volume mesh generation.

`mesh/prism/controls/`split?

Enables you to set parameters for splitting the prism layers after the initial prism layers are generated, to generate the total number of layers required. Specify the number of divisions per layer.

`mesh/prism/controls/`zone-specific-growth/

Enters the prism growth controls menu.

`mesh/prism/controls/zone-specific-growth/`apply-growth

Applies the zone-specific growth parameters specified.

`mesh/prism/controls/zone-specific-growth/`clear-growth

Clears the zone-specific growth specified.

`mesh/prism/controls/zone-specific-growth/`list-growth

Lists the zone-specific growth parameters specified for individual zones in the console.

`mesh/prism/`create

Creates prism layers on one or more boundary face zones based on the offset method, growth method, number of layers, and rate specified.

`mesh/prism/`improve/

Enters the prism improve menu.

`mesh/prism/improve/`improve-prism-cells

Collects and smooths cells in layers around poor quality cells. Cells with quality worse than the specified threshold value will be identified, and the nodes of the cells surrounding the poor quality cells will be moved to improve quality.

`mesh/prism/improve/`smooth-brute-force?

Forcibly smooths cells if cell skewness is still high after regular smoothing.

`mesh/prism/improve/`smooth-cell-rings

Specifies the number of cell rings around the skewed cell used by `improve-prism-cells`.

`mesh/prism/improve/`smooth-improve-prism-cells

Uses a combination of node movement and optimized smoothing to improve the quality. This command is a combination of the `smooth-prism-cells` and `improve-prism-cells` commands. The cell aspect ratio will also be maintained based on the value specified for `max-aspect-ratio`.

mesh/prism/improve/smooth-prism-cells

Enables optimization based smoothing of prism cells. The nodes of cells with quality worse than the specified threshold value will be moved to improve quality. The cell aspect ratio will also be maintained based on the value specified for `max-aspect-ratio`.

mesh/prism/improve/smooth-sliver-skew

Specifies the skewness above which prism cells will be smoothed.

mesh/prism/**list-parameters**

Shows all prism mesh parameters.

mesh/prism/**mark-ignore-faces**

Enables you to mark the faces to be ignored during prism meshing.

mesh/prism/**mark-nonmanifold-nodes**

Enables you to mark the non-manifold prism base nodes. A list of the non-manifold nodes will be printed in the console. The faces connected to the non-manifold nodes will also be marked. You can use this command after specifying zone-specific prism settings, prior to generating the prisms to verify that non-manifold configurations do not exist.

mesh/prism/**post-ignore/**

Contains the following options for ignoring prism cells:

mesh/prism/post-ignore/create-cavity

Creates a cavity in regions where prism quality is adequate, but the quality of adjacent tetrahedra is poor. The cavity is created based on the tetrahedral cell zone, the quality measure and the corresponding threshold value, and the additional number of cell rings specified. You can create a cavity comprising only tetrahedral cells or optionally include prism cells in the cavity created. When prism cells are also included in the cavity, you can specify whether the non-conformal interface is to be created.

mesh/prism/post-ignore/mark-cavity-prism-cap

Marks the prism cap faces and tetrahedral cell faces bounding the cavity to be created in regions where prism quality is adequate, but the quality of adjacent tetrahedra is poor. Specify the tetrahedral cell zone, the quality measure and the corresponding threshold value to be used, and the additional number of cell rings based on which the cavity will be created.

mesh/prism/post-ignore/mark-prism-cap

Marks the prism cap faces for ignoring prism cells in regions of poor quality cells and sharp corners. Specify the prism cell zone and the basis for ignoring prism cells and the relevant parameters. The prism cells can be ignored based on quality, intersection, (both enabled by default), warp, and features (both disabled by default). Specify the quality measure and threshold value to be used for ignoring cells based on quality and (if applicable) the feature edges for ignoring cells based on features. Additionally, specify whether cells are to be

marked in regions of high aspect ratio and based on feature angle, and the additional number of cell rings based on which prism cells will be removed.

mesh/prism/post-ignore/post-remove-cells

Enables you to remove prism cells in layers around poor quality cells and sharp corners. Specify the prism cell zone, the basis for ignoring prism cells (quality, intersection, warp, features) and the relevant parameters. Specify the number of cell rings to be removed around the marked cells. Cells will be marked for removal in regions of sharp corners based on quality, intersection, warp, and features (as applicable) and then extended based on the number of cell rings specified. Additional cells will be marked for removal in regions of high aspect ratio and based on feature angle (if applicable) around the exposed prism side. The boundary will be smoothed at feature corners after the prism cells have been removed. The prism-side faces exposed by the removal of the prism cells will be collected in a zone named `prism-side-#`, while for a zone `wall-n`, the faces corresponding to the ignored prism cells will be collected in a zone named `wall-n:ignore`. You can also optionally smooth the prism side nodes from the base node to the cap node to create better triangles for the non-conformal interface.

mesh/prism/quality-method

Specifies the quality method used during prism generation.

mesh/prism/reset-parameters

Resets all prism parameters.

mesh/prism/split/

Contains options for splitting the prism layers after the initial prism layers are generated, to generate the total number of layers required.

mesh/prism/split/split

Enables you to split the prism layers after the initial prism layers are generated, to generate the total number of layers required. Specify the prism cell zones to be split and the number of divisions per layer. You can also choose to use the existing growth rate (default) or specify the growth rate to be used while splitting the prism layers.

mesh/pyramid/

Enters the pyramid menu.

mesh/pyramid/controls/

Enters the pyramid controls menu.

mesh/pyramid/controls/neighbor-angle

Sets the threshold dihedral angle used to limit the neighboring faces considered for pyramid creation. For example, if the value is set to 110° and the angle between a given quadrilateral face and a neighboring triangular face is greater than 110°, the resulting pyramid will not include the triangular face.

`mesh/pyramid/controls/`offset-factor

Specifies the fraction of the computed pyramid height (offset) by which the pyramid heights will be randomly adjusted. The default value is 0, indicating that all pyramids will have the exact height computed. A value of 0.1, for example, will limit each adjustment to ±10% of the computed height.

`mesh/pyramid/controls/`offset-scaling

Specifies the scaling, to be used to determine the height of the pyramid.

`mesh/pyramid/controls/`vertex-method

Specifies the method by which the location of the new vertex of the pyramid will be determined. The `skewness` method is used by default.

`mesh/pyramid/`create

Creates a layer of pyramids on the quad face zone.

`mesh/`rapid-octree/

Enters the rapid octree menu, which provides text commands for using the Rapid Octree mesher.

`mesh/rapid-octree/`boundary-treatment

Selects the boundary treatment option. Enter `0` for the Boundary Projection treatment or `1` for the Cartesian Snapping treatment.

`mesh/rapid-octree/`bounding-box

Defines/modifies the bounding box around the geometry.

`mesh/rapid-octree/`create

Creates a mesh using the Rapid Octree mesher.

`mesh/rapid-octree/`distribute-geometry?

Enables/disables the distribution of the input geometry across partitions / compute nodes, so that it is not copied to each process. This reduces the memory requirements of the mesh generation significantly, especially for geometries with a high number of triangles. Note that this geometric distribution is enabled by default and is automatically deactivated if the geometry is not fully enclosed by the defined bounding box.

`mesh/rapid-octree/`flow-volume

Specifies the volume to be filled by the mesh.

`mesh/rapid-octree/`geometry

Allows you to apply the Rapid Octree mesher to a defined mesh object or geometry object rather than all available surface zones. Note that using a mesh object with multiple volumetric regions allows you to generate multiple disconnected cell zones that can be coupled by a non-

conformal mesh interface in the solution mode; all other input objects result in the creation of a single volume / cell zone.

mesh/rapid-octree/mesh-sizing/

Enters the mesh sizing menu, which allows you to define the cell sizes.

mesh/rapid-octree/mesh-sizing/add-surface-sizing

Adds a target mesh size for a list of surface zones. This text command is only available when the Boundary Projection treatment is selected through the `/mesh/rapid-octree/boundary-treatment` text command.

mesh/rapid-octree/mesh-sizing/boundary-cell-size

Sets the default cell size for the geometry.

mesh/rapid-octree/mesh-sizing/boundary-layers

Sets the minimum number of constant size cells adjacent to the geometry.

mesh/rapid-octree/mesh-sizing/buffer-layers

Sets the number of buffer layers, to control the transition from fine to coarse cells. Enter 0 for the fastest possible transition, or higher numbers for a slower transition; as the value increases, so too will the thickness of cell layers of the same size. The number of constant-size cell layers will be at least 2 to the power of the number you enter here.

mesh/rapid-octree/mesh-sizing/clear-all-surface-sizings

Removes all defined size functions. This text command is only available when the Boundary Projection treatment is selected through the `/mesh/rapid-octree/boundary-treatment` text command.

mesh/rapid-octree/mesh-sizing/feature-angle-refinement

Defines the angular threshold and number of refinement levels for features. This text command is only available when the Boundary Projection treatment is selected through the `/mesh/rapid-octree/boundary-treatment` text command.

mesh/rapid-octree/mesh-sizing/max-cell-size

Sets the maximum cell size in the octree mesh.

mesh/rapid-octree/refinement-regions/

Enters the rapid octree refinement region menu, which allows you to manage the refinement regions.

mesh/rapid-octree/refinement-regions/add

Adds a refinement region to the domain.

mesh/rapid-octree/refinement-regions/delete

Deletes a refinement region.

mesh/rapid-octree/refinement-regions/list

Lists all of the refinement regions.

mesh/rapid-octree/reset-bounding-box

Redefines the bounding box extents to encompass all of the surfaces of the currently selected geometry, and updates the base length scale used in the mesh generation process.

mesh/rapid-octree/undo-last-meshing-operation

Attempts to restore the object state (including its surfaces) as it was prior to the meshing operation performed by the Rapid Octree mesher.

mesh/rapid-octree/verbosity

Sets the verbosity of the messages printed by the Rapid Octree mesher.

mesh/repair-face-handedness

Reverses face node orientation.

mesh/reset-mesh

Clears the entire mesh.

mesh/reset-mesh-parameter

Resets all parameters to their default value.

mesh/scoped-prisms/

Contains options for creating scoped prism controls for mesh objects.

mesh/scoped-prisms/create

Creates a new scoped prism control based on the parameters and scope specified. Specify the name, offset method, first height or aspect ratio, number of layers, and rate or last percent. Select the mesh object and set the scope (`fluid-regions`, `named-regions`, or `solid-regions`). Specify the zones to grow prisms (`all-zones`, `only-walls`, `selected-face-zones`, or `selected-labels`, or `solid-fluid-interface`). When `named-regions` and/or `selected-face-zones` or `selected-labels` are selected, specify the volume and/or boundary scope. If interior baffle zones are selected, retain the option to grow prisms on both sides of the baffles or disable it to grow prisms on one side.

mesh/scoped-prisms/delete

Deletes the specified scoped prism control.

`mesh/scoped-prisms/`growth-options

Enables you to specify scoped prism growth options. Select `Fix First Height` if required, and specify the gap factor, maximum aspect ratio, prism quality method, and the threshold quality value for stair stepping.

`mesh/scoped-prisms/`list

Lists all the defined scoped prism controls.

`mesh/scoped-prisms/`modify

Modifies the specified control based on the parameters specified.

`mesh/scoped-prisms/`read

Reads in the specified scoped prism control file (*.pzmcontrol).

`mesh/scoped-prisms/`set-no-imprint-zones

Used to specify face zones that should not be imprinted during prism generation.

`mesh/scoped-prisms/`set-advanced-controls

Used to specify various controls for scoped prisms. Prompts include setting iterations for normal based prisms, smoothing, prism improvement, automatic node movement, and warp improvement. Prompts also include checks for stair-step interactions, as well as proximity, quality, and the exposure of quad quality. Automatic stair-stepping occurs during prism generation based on the proximity and quality limits. You can intentionally avoid stair-stepping by setting the last three prompts (proximity, quality, and the exposure of quad quality) to `no`, although you may also retain some poor quality cells.

`mesh/scoped-prisms/`write

Writes the scoped prism controls to a prism control file (*.pzmcontrol). Specify the scoped prism file name.

`mesh/`selective-mesh-check

Performs a customized mesh check on specific zones rather than all zones.

`mesh/`separate/

Separates cells by various user-defined methods.

`mesh/separate/`local-regions/

Enters the local refinement menu.

`mesh/separate/local-regions/`define

Enables you to define the parameters for the refinement region.

`mesh/separate/local-regions/delete`

Enables you to delete a refinement region.

`mesh/separate/local-regions/init`

Deletes all current regions and adds the default refinement region.

`mesh/separate/local-regions/list-all-regions`

Lists all the refinement regions.

`mesh/separate/separate-cell-by-face`

Separates cells that are connected to a specified face zone into another cell zone. This separation method applies only to prism cells.

`mesh/separate/separate-cell-by-mark`

Separates cells within a specified local region into another cell zone.

`mesh/separate/separate-cell-by-region`

Separates contiguous regions within a cell zone into separate cell zones.

`mesh/separate/separate-cell-by-shape`

Separates cells with different shapes (pyramids, tetrahedra, etc.) into separate cell zones.

`mesh/separate/separate-cell-by-size`

Separates cells based on the specified minimum and maximum cell sizes.

`mesh/separate/separate-cell-by-skew`

Separates cells based on the specified cell skewness.

`mesh/tet/`

Enters the tetrahedral mesh menu.

`mesh/tet/controls/`

Enters the tet controls menu.

`mesh/tet/controls/adv-front-method/`

Enters the advancing front refinement controls menu.

`mesh/tet/controls/adv-front-method/first-improve-params`

Defines the refining front improvement parameters for the advancing front method.

`mesh/tet/controls/adv-front-method/refine-parameters`

Defines the cell zone improvement parameters for the advancing front method.

mesh/tet/controls/adv-front-method/second-improve-params

Defines the cell zone improvement parameters for the advancing front method.

mesh/tet/controls/adv-front-method/skew-improve/

Enters the refine improve controls menu.

mesh/tet/controls/adv-front-method/skew-improve/attempts

Specifies the number of overall improvement attempts for the advancing front method.

mesh/tet/controls/adv-front-method/skew-improve/boundary-sliver-skew

Specifies the boundary sliver skewness for the advancing front method. This parameter is used for removing sliver cells along the boundary.

mesh/tet/controls/adv-front-method/skew-improve/iterations

Specifies the number of improvement iterations in each attempt for the advancing front method.

mesh/tet/controls/adv-front-method/skew-improve/sliver-skew

Specifies the sliver skewness for the advancing front method. This parameter is used for removing sliver cells in the interior.

mesh/tet/controls/adv-front-method/skew-improve/target-low-skew

Specifies the targeted skewness threshold above which cells will be improved. The improve operation will attempt to improve cells with skewness above the `target-low-skew` value specified, but there will be no attempt to reduce the skewness below the specified value. A limited set of improve operations will be used as compared to the operations required for the `target-skew` value-based improvement. The value specified could be approximately 0.1 lower than the `target-skew` value.

mesh/tet/controls/adv-front-method/skew-improve/target-skew

Specifies the targeted skewness during improvement for the advancing front method.

mesh/tet/controls/adv-front-method/skew-improve/target?

Enables you to enable targeted skewness-based refinement for the advancing front method. This option enables you to improve the mesh until the targeted skewness value is achieved.

mesh/tet/controls/cell-sizing

Specifies the cell sizing function for refinement. You can select `geometric`, `linear`, `none`, or `size-field` as appropriate.

mesh/tet/controls/compute-max-cell-volume

Computes the maximum cell volume for the current mesh.

mesh/tet/controls/delete-dead-zones?

Specifies the maximum allowable cell volume.

mesh/tet/controls/delete-unused-nodes?

Toggles the deleting of unused nodes during mesh initialization.

> **Note:**
>
> The `delete-unused-nodes?` option is no longer supported and will be re-moved at a future release.

mesh/tet/controls/improve-mesh/

Enters the improve mesh controls menu.

mesh/tet/controls/improve-mesh/improve?

Automatically improves the mesh.

mesh/tet/controls/improve-mesh/laplace-smooth

Enables you to specify the Laplace smoothing parameters.

mesh/tet/controls/improve-mesh/skewness-smooth

Enables you to specify the skewness smooth parameters.

mesh/tet/controls/improve-mesh/swap

Enables you to specify the face swap parameters.

mesh/tet/controls/improve-surface-mesh?

Enables you to improve the surface mesh by swapping face edges where Delaunay violations occur.

> **Note:**
>
> The `improve-surface-mesh?` option is no longer supported and will be removed at a future release.

mesh/tet/controls/max-cell-length

Specifies the maximum allowable cell length.

mesh/tet/controls/max-cell-volume

Specifies the maximum allowable cell volume.

mesh/tet/controls/non-fluid-type

Selects the non-fluid cell zone type. After the mesh is initialized, any non-fluid zones will be set to this type. If the mesh includes multiple regions (for example, the problem for which you are creating the mesh includes a fluid zone and one or more solid zones), and you plan to refine all of them using the same refinement parameters, modify the non-fluid type before generating the mesh.

> **Note:**
>
> For zone-based meshing, if any cell zone has at least one boundary zone type as inlet, it will automatically be set to fluid type. For object based meshing, volume region type is used to determine the cell zone type.

mesh/tet/controls/print-region-based-sizing

Displays local sizing settings (max cell length) for specified region(s).

mesh/tet/controls/refine-levels

Sets the number of refinement levels.

mesh/tet/controls/refine-method

Enables you to select the refinement method. You can select either skewness-based refinement or the advancing front method.

> **Note:**
>
> The skewness-based refinement option is no longer supported and will be removed at a future release.

mesh/tet/controls/remove-slivers/

Enters the sliver remove controls menu.

mesh/tet/controls/remove-slivers/angle

Specifies the maximum dihedral angle for considering the cell to be a sliver

mesh/tet/controls/remove-slivers/attempts

Specifies the number of attempts overall to remove slivers.

mesh/tet/controls/remove-slivers/iterations

Specifies the number of iterations to be performed for the specific sliver removal operation.

mesh/tet/controls/remove-slivers/low-skew

Specifies the targeted skewness threshold above which cells will be improved. The improve operation will attempt to improve cells with skewness above the `low-skew` value specified, but there will be no attempt to reduce the skewness below the specified value. A limited set of improve operations will be used as compared to the operations required for the `skew` value-based improvement.

mesh/tet/controls/remove-slivers/method

Enables you to select the method for sliver removal. The default method used is the `fast` method. The `fast` and the `aggressive` methods use the same controls and give similar results for good quality surface meshes. In case of poor surface meshes, the `aggressive` method will typically succeed in improving the mesh to a greater extent, but it may be slower than the `fast` method.

mesh/tet/controls/remove-slivers/remove?

Enables/disables the automatic removal of slivers.

mesh/tet/controls/remove-slivers/skew

Specifies the skewness threshold for sliver removal.

mesh/tet/controls/set-region-based-sizing

Allows you to specify local sizing settings (max cell length) for specified region(s)

mesh/tet/controls/skewness-method/

Enters the skewness refinement controls menu.

> **Note:**
>
> The `skewness-method/` menu is no longer supported and will be removed at a future release.

---

mesh/tet/controls/skewness-method/levels

Specifies the number of refinement levels for skewness-based refinement.

mesh/tet/controls/skewness-method/max-skew-improve?

Enables/disables the skewness-based improvement during refinement.

mesh/tet/controls/skewness-method/must-improve-skewness?

Enables/disables the modification of the default auto refinement parameters in order to improve skewness after refinement.

mesh/tet/controls/skewness-method/refine-boundary-cells?

Enables/disables the automatic refinement of boundary cells during refinement.

mesh/tet/controls/skewness-method/refine-cells?

Enables/disables the automatic refinement of cells during refinement.

mesh/tet/controls/skewness-method/skew-improve/

Contains options for improving refinement by the skewness method. This sub-menu is available when `max-skew-improve?` is enabled.

mesh/tet/controls/skewness-method/skew-improve/attempts

Specifies the number of overall improvement attempts for the skewness method.

mesh/tet/controls/skewness-method/skew-improve/boundary-sliver-skew

Specifies the boundary sliver skewness for the skewness method. This parameter is used for removing sliver cells along the boundary.

mesh/tet/controls/skewness-method/skew-improve/iterations

Specifies the number of improvement iterations in each attempt for the skewness method.

mesh/tet/controls/skewness-method/skew-improve/sliver-skew

Specifies the sliver skewness for the skewness method. This parameter is used for removing sliver cells in the interior.

mesh/tet/controls/skewness-method/skew-improve/target-low-skew

Specifies the targeted skewness threshold above which cells will be improved. The improve operation will attempt to improve cells with skewness above the `target-low-skew` value specified, but there will be no attempt to reduce the skewness below the specified value. A limited set of improve operations will be used as compared to the operations required for the `target-skew` value-based improvement. The value specified could be approximately 0.1 lower than the `target-skew` value.

mesh/tet/controls/skewness-method/skew-improve/target-skew

Specifies the targeted skewness during improvement for the skewness method.

mesh/tet/controls/skewness-method/skew-improve/target?

Enables you to enable targeted skewness-based refinement for the skewness method. This enables you to improve the mesh until the targeted skewness value is achieved.

mesh/tet/controls/skewness-method/smooth-mesh?

Enables/disables the automatic smoothing of the mesh during refinement.

mesh/tet/controls/skewness-method/sort-boundary-faces?

Enables/disables the automatic sorting of boundary faces by size during refinement.

mesh/tet/controls/skewness-method/sort-cells?

Enables/disables the automatic reverse sorting of cells by skewness during refinement.

mesh/tet/controls/skewness-method/swap-faces?

Enables/disables the automatic swapping of faces during refinement.

mesh/tet/controls/skewness-method/type

Enables you to select the appropriate pre-defined skewness refinement parameters. You can select `default`, `fast-transition`, or `incremental-improve` as required.

mesh/tet/controls/use-max-cell-size?

Enables you to use the maximum cell size specified instead of recomputing the value based on the objects, when the volume mesh is generated. This option is disabled by default.

mesh/tet/delete-virtual-cells

Deletes virtual cells created due to the use of the `keep-virtual-entities?` option.

mesh/tet/improve/

Enters the tet improve menu.

mesh/tet/improve/collapse-slivers

Attempts to collapse the nodes of a skewed sliver cell on any one of its neighbors.

mesh/tet/improve/improve-cells

Improves skewed tetrahedral cells.

mesh/tet/improve/refine-boundary-slivers

Attempts to increase the volume of boundary slivers to create a valid tet cell. Tetrahedra having one or two faces on the boundary are identified and then the appropriate edge split. The split node is then smoothed such that the volume of the tetrahedron increases, thereby creating a valid tet cell.

mesh/tet/improve/refine-slivers

Attempts to remove the sliver by placing a node at or near the centroid of the sliver cell. Swapping and smoothing are performed to improve the skewness. You can also specify whether boundary cells are to be refined. Refining the boundary cells may enable you to carry out further improvement options such as smoothing, swapping, and collapsing slivers.

mesh/tet/improve/skew-smooth-nodes

Applies skewness-based smoothing to nodes on the tetrahedral cell zones to improve the mesh quality.

`mesh/tet/improve/`sliver-boundary-swap

Removes boundary slivers by moving the boundary to exclude the cells from the zone.

`mesh/tet/improve/`smooth-boundary-sliver

Smooths nodes on sliver cells having all four nodes on the boundary until the skewness value is less than the specified value. The default values for the skewness threshold, minimum dihedral angle between boundary faces, and feature angle are 0.985, 10, and 30, respectively.

`mesh/tet/improve/`smooth-interior-sliver

Smooths non-boundary nodes on sliver cells having skewness greater than the specified threshold value. The default value for the skewness threshold is 0.985.

`mesh/tet/improve/`smooth-nodes

Enables you to apply either Laplacian or variational smoothing to nodes on the tetrahedral cell zones to improve the mesh quality.

`mesh/tet/improve/`swap-faces

Performs interior face swapping to improve cell skewness.

`mesh/tet/`init

Generates the initial Delaunay mesh by meshing the boundary nodes.

`mesh/tet/`init-refine

Generates the tetrahedral mesh.

`mesh/tet/`local-regions/

Enters the local refinement menu.

`mesh/tet/local-regions/`activate

Activates the specified regions for refinement.

`mesh/tet/local-regions/`deactivate

Deactivates the specified regions for refinement.

`mesh/tet/local-regions/`define

Defines the refinement region according to the specified parameters.

`mesh/tet/local-regions/`delete

Deletes the specified refinement region.

`mesh/tet/local-regions/`ideal-vol

Reports the volume of an ideal tetrahedron for the edge length specified.

`mesh/tet/local-regions/`init

Defines the default refinement region encompassing the entire geometry.

`mesh/tet/local-regions/`list-all-regions

Lists all refinement region parameters and the activated regions in the console.

`mesh/tet/local-regions/`refine

Refines the active cells inside the selected region based on the specified refinement parameters.

`mesh/tet/`preserve-cell-zone

Allows you to specify the cell zones to be preserved during the meshing process.

`mesh/tet/`refine

Refines the initialized mesh.

`mesh/tet/`trace-path-between-cells

Detects holes in the geometry by tracing the path between the two specified cells

`mesh/`thin-volume-mesh/

Creates a sweep-like mesh for a body occupying a thin gap. You define source and target boundary faces zones (the source face normal should point to the target). The source face mesh may be triangles or quads.

`mesh/thin-volume-mesh/`create

Initiates the dialog box to specify source and target faces and specify the following parameters

- `Gap Thickness`

  Helps Fluent Mesher determine the region to be remeshed. If set to 0, the mesher will automatically calculate the gap thickness. If non-zero, the `Gap Thickness` defines the maximum separation between source and target zones in the swept-mesh region (default = 0).

- `Number of Divisions`

  Specifies the number of layers created between source and target faces (default = 1).

- `Growth Rate`

  Specifies the maximum thickness ratio between two adjacent layers (default = 1).

- `Remesh Overlap Zones`

If yes, any overlapped part of the surface mesh on the target and adjacent faces will be remeshed. Original meshes are replaced (default = yes).

An example **create** dialog box:

```
Meshing/mesh/thin-volume-mesh/create
()
Source face zones(1) [()] source*
Source face zones(2) [()]
()
Target face zones(1) [()] target
Target face zones(2) [()]
Gap Thickness [0] 1
Number of Divisions [1] 5
Growth Rate [1]
Remesh Overlap Zones? [yes]
Start thin-gap mesh:
- Create and project nodes and faces on targets ...
- Project nodes on targets ...
- Create layer nodes ...
- Create side quad faces ...
- Create interior layer cells ...
- Retriangluate target face zones ...
- Retriangulate adjacent face zones ...
Done
```

mesh/**zone-names-clean-up**

Cleans up face and cell zone names by removing excessive characters so that the zone names are easier to read. During the course of creating a mesh, various mesh manipulations can cause the face and cell zone names to become cluttered (with colons, dashes, and numbers for example). Using this command, zone names are reduced to include only the original zone prefix, a single colon, and the zone's ID appended to the end.

For example, consider the following listing of zone names:

id	name	type	count	tri	quad
397	solid:11111-side-side:397	velocity-inlet	78	0	78
395	solid:11111-side	wall	52	0	52
380	solid:11111-side-side	wall	78	0	78
368	solid:11111	velocity-inlet	13	7	6
357	solid:11111-side-internal	internal	52	0	52
15	solid:11111-internal	internal	13	7	6
8	solid:1	wall	53	35	18
381	interior-381	interior	806	0	806
358	interior-358	interior	103	21	82
346	interior--solid	interior	144	111	33

Upon invoking the command:

```
Meshing/mesh> zone-names-clean-up

Cleaning up face and cell zone names...

Meshing/mesh>
```

The zone listing becomes:

id	name	type	count	tri	quad
397	solid:397	velocity-inlet	78	0	78
395	solid:395	wall	52	0	52
380	solid:380	wall	78	0	78
368	solid:368	velocity-inlet	13	7	6
357	solid:357	internal	52	0	52
15	solid:15	internal	13	7	6
8	solid:8	wall	53	35	18
381	interior-381	interior	806	0	806
358	interior-358	interior	103	21	82
346	interior--solid	interior	144	111	33

Chapter 9: `objects/`

`objects/`**cad-association/**

Contains options for modifying the selected objects based on the associated CAD entities and attaching/detaching the CAD entities from the objects. This menu is available when the **CAD Assemblies** tree is created during CAD import.

`objects/cad-association/`**attach-cad**

Attaches CAD entities to the selected geometry/mesh objects. Select the geometry/mesh objects and specify the path for the CAD entities to be associated with the objects. The selected geometry/mesh objects will be associated with the CAD entities which will then be locked.

`objects/cad-association/`**detach-all-objects**

Detaches all the CAD objects associated with the geometry/mesh objects. Specify the type of objects (`geom` or `mesh`) to be detached. All association will be removed and the geometry/mesh objects will be independent of changes to the CAD entities.

`objects/cad-association/`**detach-objects**

Detaches the CAD objects associated with the specified geometry/mesh objects. All association will be removed and the selected geometry/mesh objects will be independent of changes to the CAD entities.

`objects/cad-association/`**query-object-association**

Returns a list of the CAD entities associated with the objects selected.

`objects/cad-association/`**restore-cad**

Restores the geometry/mesh objects from the associated CAD objects.

`objects/cad-association/`**unlock-cad**

Unlocks the CAD objects associated with the selected geometry/mesh objects.

`objects/cad-association/`**update-objects**

Updates the specified geometry/mesh objects based on changes to the associated CAD objects.

`objects/cad-association/`**update-all-objects**

Updates all geometry/mesh objects based on changes to the associated CAD objects. Specify the type of objects (`geom` or `mesh`) to be updated.

objects/change-object-type

Allows you to change the object type (`geom`, or `mesh`).

objects/check-mesh

Checks the mesh on the specified objects for connectivity and orientation of faces. The domain extents, volume statistics, and face area statistics will be reported along with the results of other checks on the mesh.

objects/create

Creates the object based on the priority, cell zone type, face zone(s), edge zone(s), and object type specified. You can specify the object name or retain the default blank entry to have the object name generated automatically.

objects/create-and-activate-domain

Creates and activates the domain comprising the face zone(s) from the object(s) specified.

objects/create-groups

Creates a face group and an edge group comprising the face zone(s) and edge zone(s) included in the specified object(s), respectively.

objects/create-intersection-loops

Allows you to create intersection loops for objects.

- The `collectively` option creates an interior edge loop at the intersection between two adjacent face zones included in the same object and between multiple objects.

- The `individually` option creates an interior edge loop at the intersection between two adjacent face zones included in the same object.

objects/create-multiple

Creates multiple objects by creating an object per face zone specified. The objects will be named automatically based on the prefix and priority specified.

objects/create-new-mesh-object/

Contains options for creating a new mesh object by wrapping or remeshing existing objects.

objects/create-new-mesh-object/remesh

Creates a new mesh object by remeshing geometry objects individually or collectively.

objects/create-new-mesh-object/wrap

Creates a new mesh object by wrapping the specified objects individually or collectively.

objects/delete

Deletes the specified object(s).

`objects/`delete-all

Deletes all the defined objects.

`objects/`delete-all-geom

Deletes all the defined `geom` objects.

`objects/`delete-unreferenced-faces-and-edges

Deletes all the faces and edges that are not included in any defined objects.

`objects/`extract-edges

Extracts the edge zone(s) from the face zone(s) included in the specified object(s), based on the edge-feature-angle value specified (`/objects/set/set-edge-feature-angle`).

`objects/`improve-feature-capture

Enables you to imprint the edges comprising the object on to the object face zones to improve feature capture for mesh objects. You can specify the number of imprinting iterations to be performed.

> **Note:**
>
> The geometry objects used to create the mesh objects should be available when the `improve-feature-capture` command is invoked. Additionally, the face zones comprising the objects should be of type other than geometry.

`objects/`improve-object-quality

Enables you to improve the surface mesh quality for mesh objects. Select the mesh objects and the method for improving the surface mesh. The `smooth-and-improve` method improves the mesh by a combination of smoothing, swapping, and surface mesh improvement operations. Object normals are correctly oriented and island faces are also deleted. You can optionally coarsen the surface mesh by specifying a suitable coarsening factor. Additional imprinting operations can be done to improve feature capture on the surface mesh. The `surface-remesh` method improves the mesh by remeshing based on the current size field. Object normals are correctly oriented and island faces are also deleted.

`objects/`join-intersect/

Contains options for connecting overlapping and intersecting face zones.

`objects/join-intersect/`add-objects-to-mesh-object

Allows you to specify one or more mesh objects to be added to an existing mesh object.

`objects/join-intersect/`change-region-type

Allows you to select a cell zone type (solid, fluid or dead) for a specific region.

`objects/join-intersect/`compute-regions

Closed cell zone regions are computed from the specified mesh object. You may include a material point, if desired.

`objects/join-intersect/`controls/

`objects/join-intersect/controls/`remesh-post-intersection?

Used to enable or disable automatic post-remesh operation after join or intersect.

`objects/join-intersect/`create-mesh-object

Allows you to specify one or more mesh objects to be connected in one mesh object.

`objects/join-intersect/`delete-region

Removes a closed cell zone region and all of its face zones, except those which are shared by other regions, from the specified mesh object.

`objects/join-intersect/`intersect

Connects two intersecting face zones within specified angle and tolerance.

`objects/join-intersect/`join

Connects two overlapping face zones within specified angle and tolerance.

`objects/join-intersect/`list-regions

Lists details of region type, volume, material point, and comprising face zones for the topological regions computed for the specified mesh object.

`objects/join-intersect/`merge-regions

Specified regions are joined into a single region.

`objects/join-intersect/`rename-region

Enables you to specify a new name for a specified region.

`objects/`labels/

Contains options for creating and managing face zone labels.

`objects/labels/`add-zones

Adds the specified face zones to the existing face zone label for an object.

`objects/labels/`create

Creates a new face zone label for the specified face zones.

objects/labels/create-label-per-object

Creates a new face zone label for all the face zones in every object.

objects/labels/create-label-per-zone

Creates a new face zone label for each face zone in the object.

objects/labels/delete

Deletes the specified face zone labels.

objects/labels/label-unlabeled-zones

Creates labels for unlabeled face zones within the specified object. You can either use the object name as the label or provide your own label.

objects/labels/merge

Merges the specified face zone labels to a single label with the name specified.

objects/labels/remove-all-labels-on-zones

Removes all the face zone labels for the specified face zones. This command is applicable to geometry objects only.

objects/labels/remove-zones

Removes the specified face zones from the existing face zone label for an object.

objects/labels/rename

Renames the specified face zone label.

objects/list

Lists details such as cell zone type, priority, object type, comprising face and edge zones, and object reference point for all the defined objects.

objects/merge

Merges the specified objects into a single object.

objects/merge-edges

Merges all the edge zones in an object into a single edge zone.

> **Note:**
>
> If the object is composed of edge zones of different types (boundary and interior), the edge zones of the same type (boundary or interior) will be merged into a single edge zone.

objects/merge-nodes

Merges the free nodes at the object level based on the specified tolerance or using a tolerance that is a specified percentage of shortest connected edge length.

objects/merge-voids

Allows you to merge voids in the mesh object after the sewing operation.

objects/merge-walls

Merges all the face zones of type `wall` in an object into a single face zone.

objects/remove-gaps/

Contains options for removing gaps between the mesh objects specified or removing the thickness in the mesh objects specified.

objects/remove-gaps/ignore-orientation?

Allows you to set whether the orientation of the normals should be taken into account while identifying the gap to be removed.

objects/remove-gaps/remove-gaps

Allows you to remove gaps between the mesh objects specified or remove the thickness in the mesh objects specified. Select the appropriate repair option and specify the other parameters required.

objects/remove-gaps/show-gaps

Marks the faces at the gap between mesh objects based on the gap distance and percentage margin specified.

objects/rename-cell-zone-boundaries-using-labels

Renames the boundaries of the cell zones based on the existing face zone labels. This allows for the cell zone boundaries in solution mode to have names corresponding to the face zone labels in meshing mode.

> **Note:**
>
> This command will not work if you read in a volume mesh generated in a version prior to release 16.2. In such cases, regenerate the volume mesh before using the command.

objects/rename-object

Allows you to rename a specified geometry or mesh object with another specified name.

objects/rename-object-zones

Renames the face and edge zones comprising the object based on the object name. You can also specify the separator to be used.

156

objects/**restore-faces**

Restores the mesh object surface mesh from the backup created. The current mesh object face zones and cell zones will be deleted.

If the object backup is disabled (`/mesh/auto-mesh-controls/backup-object no`), you will not be able to restore the surface mesh using this command.

> **Note:**
>
> There may be a difference in the initial volume mesh generated for an object and that generated after restoring the object surface mesh due to differences in the order of zones/entities processed during volume meshing.

objects/**rotate**

Rotates the object(s) based on the angle of rotation, pivot point, and axis of rotation specified.

objects/**scale**

Scales the object(s) based on the scale factors specified.

objects/**separate-faces-by-angle**

Separates the face zone(s) comprising the object based on the angle specified.

objects/**separate-faces-by-seed**

Separates the face zone(s) comprising the object based on the seed face specified.

objects/**set/**

Contains options for setting additional object-related settings.

objects/set/**set-edge-feature-angle**

Sets the edge feature angle to be used for extracting edge zone(s) from the face zone(s) included in the object(s).

objects/set/**show-edge-zones?**

Displays the edge zone(s) comprising the object(s) drawn in the graphics window.

objects/set/**show-face-zones?**

Displays the face zone(s) comprising the object(s) drawn in the graphics window.

`objects/`**`sew/`**

Contains options related to the object sewing operation.

Note:

This menu is no longer supported, and will be removed in a future release.

`objects/sew/`**`sew`**

Connects the mesh objects to generate the conformal surface mesh.

`objects/sew/`**`set/`**

Contains additional options related to the object sewing operation.

`objects/sew/set/`**`include-thin-cut-edges-and-faces`**

Allows better recovery of thin region configurations during the sewing operation.

`objects/sew/set/`**`process-slits-as-baffles?`**

Allows you to collapse the nearly overlapping surfaces corresponding to the baffle when the sew operation is performed to create the mesh object. Specify the maximum slit thickness relative to the minimum size specified and the parallel face angle between the faces comprising the slit when process-slits-as-baffles is enabled.

`objects/sew/set/`**`zone-name-prefix`**

Allows you to specify a prefix for the zones included in the mesh object created using the sew operation.

`objects/`**`summary`**

Allows you to obtain a summary of a specified geometry or mesh object, or obtain a summary of all geometry or mesh objects.

`objects/`**`translate`**

Translates the object(s) based on the translation offsets specified.

`objects/`**`update`**

Allows you to update the objects defined when the face and/or edge zone(s) comprising the object have been deleted.

`objects/`**`volumetric-regions/`**

Contains options for manipulating volumetric regions and generating the volume mesh.

`objects/volumetric-regions/`**`auto-fill-volume`**

Creates the volume mesh for the selected volumetric regions based on the meshing parameters set.

---

objects/volumetric-regions/change-type

Enables you to change the region type.

objects/volumetric-regions/compute

Computes the volumetric regions based on the face zone labels. You can choose to use existing material points for computing the regions.

> **Note:**
>
> When regions are computed, region names and types will be based on the face zone labels of the mesh object selected. If regions are recomputed, all previous region names and types will be over written.

objects/volumetric-regions/delete

Deletes the specified volumetric regions.

> **Tip:**
>
> Deleting regions may cause face zones to be deleted. It is recommended that the region type be changed to dead instead of deleting the region.

objects/volumetric-regions/delete-cells

Deletes the cell zones of the specified regions.

objects/volumetric-regions/hexcore/

Contains options for setting hexcore mesh controls. See mesh/ (p. 105).

objects/volumetric-regions/list

Prints region information to the console, including type, volume, material point and face zones.

objects/volumetric-regions/merge

Merges specified regions in to a single region.

> **Note:**
>
> If there are shared face zones, merging regions will delete the shared face zones. However, if there are cell zones associated with the regions, then merging the regions will not delete the shared face zones. In this case, the shared face zones will be deleted when the cell zones are deleted.

objects/volumetric-regions/rename

Renames the region.

`objects/volumetric-regions/scoped-prism/`

Contains options for setting scoped prism controls.

`objects/volumetric-regions/tet/`

Contains options for setting tetrahedral mesh controls. See `mesh/` (p. 105).

`objects/volumetric-regions/update`

Recomputes the selected volumetric region(s) while preserving the region name(s) and type(s).

`objects/wrap/`

Contains options related to the object wrapping operation.

`objects/wrap/check-holes`

Allows you to check for holes in the objects. The number of hole faces marked will be reported.

`objects/wrap/recover-periodic-surfaces/`

Allows you to reestablish the periodic relationship between leader and shadow face zones on the mesh object. You will be prompted for the method and to identify the periodic face zones to be recovered.

`objects/wrap/recover-periodic-surfaces/auto`

Performs the automatic periodic recovery method using either the leader or shadow periodic face zones. Periodic recovery will be attempted in both directions without additional prompting.

`objects/wrap/recover-periodic-surfaces/manual`

Performs the manual periodic recovery method. You will be prompted for periodic and shadow face zones, and for the periodic source. The periodic source may be from an underlying periodic geometry surface or manual entry of the rotational periodic parameters (angle, origin, axis).

> **Note:**
>
> Only rotational periodicity is supported, translational periodicity is not supported currently.

`objects/wrap/set/`

Contains additional options related to the object wrapping operation.

`objects/wrap/set/add-geometry-recovery-level-to-zones`

Enables you to set the geometry recovery level (`high` or `low`) for the specified face zones.

objects/wrap/set/include-thin-cut-edges-and-faces

Allows better recovery of thin region configurations during the object wrapping operation.

objects/wrap/set/list-zones-geometry-recovery-levels

Lists the zones based on geometry recovery level specified.

objects/wrap/set/max-free-edges-for-hole-patching

Allows you to set the maximum number of free edges in a loop to fill the holes.

objects/wrap/set/minimum-relative-topo-area

Specifies the minimum relative topological area for shrink wrapping.

objects/wrap/set/minimum-relative-topo-count

Specifies the minimum relative topological count for shrink wrapping.

objects/wrap/set/minimum-topo-area

Specifies the minimum topological area for shrink wrapping.

objects/wrap/set/minimum-topo-count

Specifies the minimum topological count for shrink wrapping.

objects/wrap/set/relative-feature-tolerance

Specifies the relative feature tolerance for shrink wrapping.

objects/wrap/set/report-holes?

Allows you to check for holes in the mesh object created. Holes, if any will be reported at the end of the object wrapping operation.

objects/wrap/set/resolution-factor

Sets the resolution factor for shrink wrapping. This option can be used to set sampling coarser or finer than the final surface mesh.

objects/wrap/set/shrink-wrap-rezone-parameters

Allows you to set the parameters for improving the mesh object surface quality using rezoning. The geometry object zones will be separated based on the separation angle specified to improve the feature imprinting on the mesh object.

objects/wrap/set/zone-name-prefix

Allows you to specify a prefix for the zones included in the mesh object created using the object wrapping operation.

objects/wrap/**wrap**

Creates the mesh objects based on the geometry objects selected and other object wrapping parameters specified.

Chapter 10: `parallel/`

`parallel/agglomerate`

Recombines distributed mesh data into a single partition on compute node 0.

`parallel/auto-partition?`

Automatically partitions face-zones for parallel meshing.

`parallel/distribute`

Allocates mesh to the compute nodes based on the computed partitions.

`parallel/print-partition-info`

Displays computed partition data to the console.

`parallel/spawn-solver-processes`

Specifies the number of solver processes. Additional processes will be spawned as necessary when switching to solution mode in Linux with the default MPI. You will also be prompted for (Linux and mixed Windows/Linux) interconnect type, machine list or host file, and (Linux and mixed Windows/Linux) option to be used.

`parallel/thread-number-control`

Controls the maximum number of threads on each machine.

Chapter 11: `preferences/`

Important:

Values that correspond to drop-down entries in the GUI version of **Preferences** are case-sensitive must be entered with the same capitalization as the GUI version and be contained within quotation marks.

`preferences/`**appearance/**

Enter the menu for preferences covering appearance.

`preferences/appearance/`**ansys-logo/**

Enter the menu for controlling Ansys logo visibility.

`preferences/appearance/ansys-logo/`**visible**

Enable or disable the visibility of the Ansys logo in the graphics window.

`preferences/appearance/ansys-logo/`**color**

Specify whether the Ansys logo is white or black.

`preferences/appearance/`**axis-triad**

Enable or disable the visibility of the axis triad in the graphics window.

`preferences/appearance/`**charts/**

Enter the menu for controlling the display of 2D charts/plots.

`preferences/appearance/charts/`**curve-colors**

Specify the initial set of default colors for the rendering of curves. *Note that changing this setting requires any plots to be replotted before you see the effect of the new setting.*

`preferences/appearance/charts/`**modern-plots-enabled**

Enables enhanced plots, which is a beta feature. Enabling this feature exposes new fields (all beta functionality).

`preferences/appearance/`**color-theme**

Specify a color theme for the appearance of Ansys Fluent.

`preferences/appearance/`completer

Enable/disable the console automatic-completer, which suggests available commands as you type in the console.

`preferences/appearance/`graphics-background-color1

Controls the primary background color of the graphics window.

`preferences/appearance/`graphics-background-color2

Controls the secondary background color when the style is set as a gradient.

`preferences/appearance/`graphics-background-style

Specify whether the background color is uniform or if there is a gradient.

`preferences/appearance/`graphics-background-theme

Provides predefined color and style selections for the graphics window background color.

`preferences/appearance/`graphics-foreground-color

Specify the color of graphics window text.

`preferences/appearance/`graphics-view

Specify whether the default view is orthographic or perspective.

`preferences/appearance/`graphics-wall-face-color

Set the default face color for when the mesh is displayed.

`preferences/appearance/`group-by-tree-view

Specify how boundary conditions are grouped in the tree.

`preferences/appearance/`number-of-files-recently-used

Controls how many recently-used files are listed in the **File** ribbon tab and the Fluent Launcher.

`preferences/appearance/`quick-property-view

Enables/Disables the "quick-edit" properties panels that appear when you select a boundary in the graphics windows.

`preferences/appearance/`ruler

Adds or removes the ruler from the graphics window. *Note that you must be in **orthographic** view for the ruler to be visible in the graphics window.*

`preferences/appearance/`selections/

Enters the menu for controlling selections in the graphics window.

preferences/appearance/selections/highlight-edge-color

Specifies the color used to highlight edges when the **Hover-Over Highlight** feature is enabled (`mouse-over-highlight-enabled`).

preferences/appearance/selections/highlight-edge-weight

Specifies the thickness of the edge highlights when the **Hover-Over Highlight** feature is enabled (`mouse-over-highlight-enabled`).

preferences/appearance/selections/highlight-face-color

Specify which color indicates that a face is selected.

preferences/appearance/selections/highlight-transparency

Specify the transparency of the coloring on a selected surface. 0.1 is fully opaque and 1 is fully transparent.

preferences/appearance/selections/mouse-hover-probe-enabled

Enable/disable the display of result values in the graphics window upon mouse-over, when graphics objects displaying solution results are displayed.

preferences/appearance/selections/mouse-over-highlight-enabled

Enable/disable the highlighted outline of a surface when hovered-over. Note that objects must be redisplayed after changing this setting before the new setting is visible.

preferences/appearance/show-model-edges

Enable/disable whether mesh edges are shown in a mesh display.

preferences/appearance/startup-page

Enable/disable the display of the startup page when Ansys Fluent is started without loading a mesh or case file.

preferences/appearance/surface-specularity

Specify the specularity of all surfaces except those included in contour plots. Sepecularity is the reflectiveness of a surface; higher values (closer to 1) equate to a more reflective surface.

preferences/appearance/surface-specularity

Specify the specularity of contour surfaces. Sepecularity is the reflectiveness of a surface; higher values (closer to 1) equate to a more reflective surface.

preferences/appearance/titles

Enable/disable the display of solver information in the graphics window.

preferences/general/

Enter the menu for general preferences.

preferences/general/automatic-transcript

Enable/disable the automatic creation of a transcript file for each Ansys Fluent session.

preferences/general/default-ioformat

Specify the default file format for saving mesh, case, and data files.

preferences/general/idle-timeout

Specify the default file format for saving case and data files.

preferences/general/warn-default-iochange

Enable/disable the warning message that appears when using case and data files saved in the legacy format.

preferences/graphics/

Enter the menu for preferences covering appearance.

preferences/graphics/animation-option

Specify whether the entire model or just a wireframe is shown during manipulations in the graphics window.

preferences/graphics/boundary-markers/

Enter the boundary markers menu.

preferences/graphics/boundary-markers/color-option

Specify whether boundary markers are a fixed color or if they match the color of the surface they are identifying.

preferences/graphics/boundary-markers/enabled

Enable/disable boundary marker display.

preferences/graphics/boundary-markers/inlet-color

Specify the color of the inlet boundary markers.

preferences/graphics/boundary-markers/marker-fraction

Specify marker density factor (0.1-1).

preferences/graphics/boundary-markers/marker-limit

Specify the maximum number of markers at each boundary.

preferences/graphics/boundary-markers/outlet-color

Specify the color of the outlet boundary markers.

preferences/graphics/boundary-markers/scale-marker

Specify the scale factor for the boundary markers (0.1-10), which controls the overall size of the markers.

preferences/graphics/boundary-markers/show-inlet-markers

Enable/disable the display of boundary markers for inlets.

preferences/graphics/boundary-markers/show-outlet-markers

Enable/disable the display of boundary markers for outlets.

preferences/graphics/colormap-settings/

Enter the colormap settings menu.

preferences/graphics/colormap-settings/alignment

Specify the default colormap location.

preferences/graphics/colormap-settings/aspect-ratio-when-vertical

Specify the length vs. width ratio for a vertical colormap, which controls the thickness of the colormap; smaller values mean a thicker colormap.

preferences/graphics/colormap-settings/border-style

Specify how/when the colormap border appears.

preferences/graphics/colormap-settings/colormap

Choose the default colormap.

preferences/graphics/colormap-settings/labels

Specify whether there is a label for every colormap value or if some are skipped.

preferences/graphics/colormap-settings/levels

Specify the default colormap size.

preferences/graphics/colormap-settings/log-scale

Enable/disable the use of a logarithmic scale for the colormap.

preferences/graphics/colormap-settings/major-length-to-screen-ratio-when-vertical

Choose the length of the colormap as a fraction of graphics window height, when the colormap is vertical.

preferences/graphics/colormap-settings/number-format-precision

Specify the colormap number label precision.

preferences/graphics/colormap-settings/number-format-type

Specify how colormap numbers are displayed.

preferences/graphics/colormap-settings/show-colormap

Enable/disable the display of colormaps.

preferences/graphics/colormap-settings/skip-value

Specify how many number labels are skipped in the colormap.

preferences/graphics/colormap-settings/text-behavior

Specify whether colormap label text automatically scales with the colormap size.

preferences/graphics/colormap-settings/text-font-automatic-vertical-size

Specify the initial font size as a ratio of the colormap overall size, for vertically aligned colormaps.

preferences/graphics/colormap-settings/text-font-fixed-size

Set the font size for colormap labels.

preferences/graphics/colormap-settings/text-font-name

Set the font for colormap labels.

preferences/graphics/colormap-settings/type

Specify whether the colormap appearance is smooth or banded.

preferences/graphics/double-buffering

Enable/disable double-buffering, which reduces screen flicker, but may use more memory on some machines.

preferences/graphics/graphics-effects/

Enter the graphics effects menu.

preferences/graphics/graphics-effects/anti-aliasing

Enable/disable the smoothing of lines and text.

preferences/graphics/graphics-effects/grid-color

Specify the color of the grid lines when the ground plane grid is shown.

preferences/graphics/graphics-effects/grid-plane-enabled

Enable/disable the display of the ground plane grid.

preferences/graphics/graphics-effects/grid-plane-offset

Set the grid plane offset from the model as a percentage of the model size.

preferences/graphics/graphics-effects/plane-direction

Specify the direction of the plane for the ground plane grid and reflections.

preferences/graphics/graphics-effects/reflections-enabled

Enable/disable model reflections (mirror-type reflections).

preferences/graphics/graphics-effects/shadow-map-enabled

Enable/disable dynamic shadows, which show shadows of geometric entities on other objects based on lighting and object orientation.

preferences/graphics/graphics-effects/show-edge-reflections

Enable/disable the display of model edges in reflections. Note that this can negatively affect performance.

preferences/graphics/graphics-effects/simple-shadows-enabled

Enable/disable the display of static shadows on the ground plane.

preferences/graphics/graphics-effects/update-after-mouse-release

Enable/disable the updating of graphics effects as a model is being manipulated in the graphics window.

preferences/graphics/graphics-window-line-width

Specify the thickness of lines that appear in the graphics window.

preferences/graphics/graphics-window-point-symbol

Specify the symbol used for indicating points in the graphics window (like the points in an XY plot).

preferences/graphics/hardcopy-settings/

Enter the menu for saving picture settings.

preferences/graphics/hardcopy-settings/hardcopy-driver

Specify the default format for saving pictures.

preferences/graphics/hardcopy-settings/hardcopy-line-width

Specify the thinkness of lines for saved pictures.

preferences/graphics/hardcopy-settings/hardcopy-image-accel

Enable/disable the use of hardware acceleration when saving images, where possible.

preferences/graphics/hidden-surface-removal-method

Specify the method for removing hidden surfaces. These methods vary in speed and quality, depending on your machine.

preferences/graphics/lighting/

Enter the lighting menu.

preferences/graphics/lighting/headlight

Turn the headlight on or off or set it as automatic.

preferences/graphics/lighting/headlight-intensity

Specify the intensity of the headlight.

preferences/graphics/lighting/lighting-method

Specify the default lighting method.

preferences/graphics/meshing-mode/

Enter the menu for meshing-specific graphics settings.

preferences/graphics/meshing-mode/graphics-window-display-timeout

Enable/disable graphics window display timeout.

preferences/graphics/meshing-mode/graphics-window-display-timeout-value

Specify the graphics window display timeout value.

preferences/graphics/performance/

Enter the menu for selecting the predefined graphics effects settings.

preferences/graphics/performance/fast-display-mode/

Enter the menu for controlling fast graphics performance preferences.

preferences/graphics/performance/fast-display-mode/faces-shown

Specify whether or not faces are shown when interacting with the model.

preferences/graphics/performance/fast-display-mode/nodes-shown

Specify whether or not nodes are shown when interacting with the model.

preferences/graphics/performance/fast-display-mode/perimeter-edges-shown

Specify whether or not the perimeter edges are shown when interacting with the model.

preferences/graphics/performance/fast-display-mode/silhouette-shown

Specify whether or not the silhouette is shown when interacting with the model.

172

preferences/graphics/performance/fast-display-mode/status

Specify whether the fast interactive display settings are on, off, or automatic.

preferences/graphics/performance/fast-display-mode/transparency

Specify whether or not transparencies remain when interacting with the model.

preferences/graphics/performance/minimum-frame-rate/

Enter the menu for minimum frame-rate settings.

preferences/graphics/performance/minimum-frame-rate/dynamic-adjustment

Enable/disable dynamic adjustment of quality loss per frame to get to the desired frame rate.

preferences/graphics/performance/minimum-frame-rate/enabled

Enable/disable minimum frame rate.

preferences/graphics/performance/minimum-frame-rate/maximum-culling-threshold

With minimum frame rate enabled, Fluent will not cull beyond this number of pixels.

preferences/graphics/performance/minimum-frame-rate/minimum-culling-threshold

With minimum frame rate enabled, Fluent will cull at least this number of pixels.

preferences/graphics/performance/minimum-frame-rate/target-fps

Specify the target frames-per-second.

preferences/graphics/performance/optimize-for

Choose a preset selection for how graphics are displayed.

preferences/graphics/point-tool-size

Specify the size of the point tool (10-100).

preferences/graphics/show-hidden-surfaces

Enable/disable the display of hidden surfaces.

preferences/icing/

Enter the menu for preferences covering the Fluent Icing workspace.

preferences/icing/advanced-flag

Enable/disable the exposure of advanced settings.

preferences/icing/beta-flag

Enable/disable the exposure of beta features.

preferences/icing/display-mesh-after-case-load

Specify which boundaries are displayed by default after the mesh is read.

preferences/icing/use-launcher

Enable/disable the use of the Fluent Launcher for starting the Fluent solver when connecting to or creating a new **Simulation**. This allows you to launch calculations using a different number of CPUs, a different machine, or a queuing system..

preferences/meshing-workflow/

Enter the menu for preferences covering the Fluent Meshing workflows.

preferences/meshing-workflow/checkpointing_option

Specify how Fluent Meshing will save data when you edit a task.

preferences/meshing-workflow/draw-settings/

Enter the menu for specifying drawing settings.

preferences/meshing-workflow/draw-settings/auto-draw

Enable/disable the automatic display of changes in the graphics window based on the current task.

preferences/meshing-workflow/draw-settings/face-zone-limit

Specify the cutoff number of face zones, beyond which, Fluent Meshing will not automatically display changes.

preferences/meshing-workflow/draw-settings/facet-limit

Specify the cutoff number facets, beyond which, Fluent Meshing will not automatically display changes.

preferences/meshing-workflow/save-checkpoint-files

Enable/disable the saving of task editing data when writing a mesh file.

preferences/meshing-workflow/temp-folder

Specify a temporary location to hold generated mesh files. If nothing is specified, Fluent Meshing will write to `%TEMP%` on Windows and to `/tmp` on Linux.

preferences/meshing-workflow/verbosity

Enable/disable the printing of additional information and messages in the Console.

preferences/navigation/

Enter the menu for controlling navigation in Ansys Fluent.

preferences/navigation/mouse-mapping/

Enable/disable the printing of additional information and messages in the Console.

preferences/navigation/mouse-mapping/additional/

Enter the menu for controlling mouse mappings that include a modifier button such as **Ctrl** and **Shift**.

preferences/navigation/mouse-mapping/additional/ctrllmbclick

Specify the action/behavoir for **Ctrl** + left-mouse-button + click.

preferences/navigation/mouse-mapping/additional/ctrllmbdrag

Specify the action/behavior for **Ctrl** + left-mouse-button + drag.

preferences/navigation/mouse-mapping/additional/ctrlmmbclick

Specify the action/behavior for **Ctrl** + middle-mouse-button + click.

preferences/navigation/mouse-mapping/additional/ctrlmmbdrag

Specify the action/behavior for **Ctrl** + middle-mouse-button + drag.

preferences/navigation/mouse-mapping/additional/ctrlrmbclick

Specify the action/behavior for **Ctrl** + right-mouse-button + click.

preferences/navigation/mouse-mapping/additional/ctrlrmbdrag

Specify the action/behavior for **Ctrl** + right-mouse-button + drag.

preferences/navigation/mouse-mapping/additional/mouseprobe

Specify whether the probe action provides a long description or a short description.

preferences/navigation/mouse-mapping/additional/mousewheel

Specify the action/behavior of the mouse-wheel.

preferences/navigation/mouse-mapping/additional/mousewheelsensitivity

Specify the sensitivity of the mouse-wheel (0 is least sensitive, 1 is most sensitive).

preferences/navigation/mouse-mapping/additional/reversewheeldirection

Reverse the behavior of the mouse-wheel.

preferences/navigation/mouse-mapping/additional/shiftlmbclick

Specify the action/behavior for **Shift** + left-mouse-button + click.

preferences/navigation/mouse-mapping/additional/shiftlmbdrag

Specify the action/behavior for **Shift** + left-mouse-button + drag.

preferences/navigation/mouse-mapping/additional/shiftmmbclick

Specify the action/behavior for **Shift** + middle-mouse-button + click.

preferences/navigation/mouse-mapping/additional/shiftmmbdrag

Specify the action/behavior for **Shift** + middle-mouse-button + drag.

preferences/navigation/mouse-mapping/additional/shiftrmbclick

Specify the action/behavior for **Shift** + right-mouse-button + click.

preferences/navigation/mouse-mapping/additional/shiftrmbdrag

Specify the action/behavior for **Shift** + right-mouse-button + drag.

preferences/simulation/

Enter the menu for simulation preferences.

preferences/simulation/flow-model

Specify the default turbulence model (viscous).

preferences/simulation/local-residual-scaling

Use local residual scaling for solution residuals.

preferences/simulation/report-definitions/

Enter the menu for report definition preferences.

preferences/simulation/report-definitions/automatic-plot-file

New report definitions will automatically create associated report files and plots.

176

Chapter 12: `report/`

`report/`**boundary-cell-quality**

> Reports the number and quality limits of boundary cells containing the specified number of boundary faces. If you specify zero for `number of boundary faces`, you will be prompted for `number of boundary nodes`.

`report/`**cell-distribution**

> Reports the distribution of cell quality or size based on the bounding limits and number of partitions specified.

`report/`**cell-quality-limits**

> Reports the cell quality limits.

`report/`**cell-size-limits**

> Reports the cell size limits.

`report/`**cell-zone-at-location**

> Returns the cell zone at or closest to the specified location.

`report/`**cell-zone-volume**

> Reports the volume of the specified cell zone.

`report/`**edge-size-limits**

> Reports the edge size limits.

`report/`**face-distribution**

> Reports the distribution of face quality or size based on the bounding limits and number of partitions specified.

`report/`**face-node-degree-distribution**

> Reports the distribution of boundary faces based on face node degree. The node degree is the number of faces connected to the node. Specify the list of boundary face zones and the minimum and maximum face node degree to be reported. You can also consider only internal nodes, if required.

`report/`**face-quality-limits**

> Reports the face quality limits.

report/face-size-limits

Reports the face size limits.

report/face-zone-area

Reports the area of the specified face zone.

report/face-zone-at-location

Reports the face zone at the given location.

report/list-cell-quality

Reports a list of cells with the specified quality measure within a specified range. The valid prefixes are `bn` (boundary node), `n` (node), `bf` (boundary face), `f` (face), and `c` (cell).

report/memory-usage

Reports the amount of memory used for all nodes, faces, and cells, and the total memory allocated.

report/mesh-size

Reports the number of nodes, faces, and cells in the mesh.

report/mesh-statistics

Writes mesh statistics (such as zone information, number of cells, faces, and nodes, range of quality and size) to an external file.

report/neighborhood-quality

Reports the maximum skewness, aspect ratio, or size change of all cells using a specified node.

report/number-meshed

Reports the number of elements that have been meshed.

report/print-info

Prints information about individual components of the mesh. This command also appears in the `boundary` menu. When you use this command, you will be prompted for an "entity" (that is, a node, face, or cell). An entity name consists of a prefix and an index. For a description of the displayed information see Reporting Mesh Information in the *Fluent User's Guide*

report/quality-method

Specifies the method to be used for reporting face and cell quality.

report/unrefined-cells

Reports the number of cells that have not been refined.

`report/`update-bounding-box

Updates the bounding box.

`report/`verbosity-level

Specifies how much information should be displayed during mesh initialization, refinement and other operations. Changing the value to 2 from the default value of 1 will produce more messages, while changing it to 0 will disable all messages.

Chapter 13: `scoped-sizing/`

`scoped-sizing/`compute

Computes the size field based on the defined size functions and/or scoped size controls.

`scoped-sizing/`create

Defines the scoped size based on the specified parameters.

`scoped-sizing/`delete

Deletes the specified scoped size controls.

`scoped-sizing/`delete-all

Deletes all the defined scoped size controls.

`scoped-sizing/`delete-size-field

Deletes the current size field.

`scoped-sizing/`list

Lists all the defined scoped size controls and the corresponding parameter values defined.

`scoped-sizing/`list-zones-uncovered-by-controls

Lists the zones for which no scoped sizing controls have been defined.

`scoped-sizing/`modify

Modifies the scoped size control definition.

`scoped-sizing/`read

Enables you to read in a scoped sizing file (*.szcontrol).

`scoped-sizing/`validate

Validates the scoped sizing controls defined. An error will be reported if the scoped sizing controls do not exist or the scope for one (or more) controls is invalid.

`scoped-sizing/`write

Enables you to write a scoped sizing file (*.szcontrol).

Chapter 14: `size-functions/`

Size functions can be defined using the commands in the `size-functions` menu. The command `size-functions/create` enables you to define size functions. You need to specify the size function type, whether face or edge zones are to be used, the boundary or edge zones for which the size function is to be defined, the size function name, and the parameters relevant to the size function type. You can also have the name generated automatically by retaining the default entry for the size function name. The size function name will be assigned based on the zone type (face or edge) and the size function type (for example, the size function **face-curvature-sf-5** indicates that the curvature size function is defined for face zones. The size function ID is 5.)

An example is

```
size-functions/create curvature face wall:x wall:y, curv-size-function 0.01 0.1 1.2 5
```

Where, `curvature` is the size function type, `face` indicates that face zones are to be used, `wall:x`, `wall:y` indicate the boundary zones for which the size function is defined, and `curv-size-function` is the size function name. The remaining values correspond to the parameters for the curvature size function, that is, minimum and maximum size, growth rate, and normal angle.

Note:

All boundary face zones and edge zones included in the global domain are available for defining size functions, even if a local domain has been activated.

`size-functions/`compute

Computes the size function based on the defined parameters.

`size-functions/`contours/

Contains options for managing contours.

`size-functions/contours/`draw

Displays contours in the graphics window. Run `compute` prior to `contours/draw`.

`size-functions/contours/`set/

Contains options to manage the contour size.

`size-functions/contours/set/`refine-facets?

Allows you to specify smaller facets if the original are too large. Default is `no`.

size-functions/create

Defines the size function based on the specified parameters.

size-functions/create-defaults

Creates default size functions based on face and edge curvature and proximity.

size-functions/delete

Deletes the specified size function or the current size field.

size-functions/delete-all

Deletes all the defined size functions.

size-functions/disable-periodicity-filter

Removes periodicity from the size field.

size-functions/enable-periodicity-filter

Applies periodicity to the size field.

Specify the angle, pivot, and axis of rotation to set up periodicity.

If periodicity has been previously defined, the existing settings will be applied.

> **Note:**
>
> Only rotational periodicity is supported, translational periodicity is not supported currently.

size-functions/list

Lists all the defined size functions and the corresponding parameter values defined.

size-functions/reset-global-controls

Resets the global controls to their default values.

size-functions/set-global-controls

Sets the values for the global minimum and maximum size, and the growth rate.

> **Note:**
>
> If you set the global minimum size to a value greater than the local minimum size defined for existing proximity, curvature, or hard size functions, a warning will appear, indicating that the global minimum size cannot be greater than the specified local minimum size.

`size-functions/`set-prox-gap-tolerance

Sets the tolerance relative to minimum size to take gaps into account. Gaps whose thickness is less than the global minimum size multiplied by this factor will not be regarded as a proximity gap.

`size-functions/`set-scaling-filter

Allows you specify the scale factor, and minimum and maximum size values to filter the size output from the size field.

`size-functions/`triangulate-quad-faces?

Identifies the zones comprising non-triangular elements and uses a triangulated copy of these zones for computing the size functions.

`size-functions/`use-cad-imported-curvature?

Allows you to disable curvature data from the nodes of the CAD facets.

Chapter 15: `switch-to-solution-mode`

`switch-to-solution-mode`

Enables you to transfer the mesh data from meshing mode to solution mode in Ansys Fluent. When you use the `switch-to-solution-mode` command, you will be asked to confirm that you want to switch to solution mode.

Appendix A. Query and Utility Functions

The following sections describe the query and utility functions available:

A.1. List Queries and Utility Functions

When performing a series of operations, you may want to list all the zones of a particular type, group, or containing a particular text string or name pattern. You can also query zones through the TUI based on a regular expression and a specified variable. You can:

- Identify face/cell zones closest to a specified location.

- Identify zones/objects of a specific type or containing a particular text string. You can also identify zones belonging to a group or object.

- Obtain object region and face zone label lists.

- Obtain mesh setup information such as zones with prism mesh settings or prism controls defined

- Use the utility functions to determine the zones created during a particular operation by performing Boolean operations on lists returned by specific functions.

- Use the `eval-expr` function to evaluate the lists returned by the functions in order that they can be used as input in the text user interface commands.

Note:

These functions are not supported in a distributed parallel environment.

Feature		Description	Utility Function
Zone query	By location	Return face zone at or closest to a specified location. **Note:** This function is not applicable to polyhedra meshes.	`(get-face-zone-at-location '(x y z))`
		Return cell zone at or closest to a specified location	`(get-cell-zone-at-location '(x y z))`
Zone lists	By zone type	Return a list of zones of the specified default zone type	`(get-zones-of-type 'type)`
	By default group	Return a list of zones of the specified default zone group or user-defined group	`(get-zones-of-group 'group)`
	By filter string	Return a list of zones whose names contain the specified filter string	`(get-face-zones-of-filter 'filter)`
			`(get-cell-zones-of-filter 'filter)`
			`(get-edge-zones-of-filter 'filter)`
			`(get-node-zones-of-filter 'filter)`
	By object	Return a list of face zones in the specified objects	`(get-face-zones-of-objects '(object-list))`
		Return a list of face zones by ID in the specified object	`(tgapi-util-get-face-zone-id-list-of-object 'object)`
			`(tgapi-util-get-face-zone-id-list-of-object "object")`
		Return a list of face zones by ID in the specified regions of an object	`(tgapi-util-get-face-zone-id-list-of-regions 'object '(region-list))`
			`(tgapi-util-get-face-zone-id-list-of-regions "object" '(region-list))`
		Return a list of edge zones in the specified objects	`(get-edge-zones-of-objects '(object-list))`
		Return a list of edge zones by ID in the specified object	`(tgapi-util-get-edge-zone-id-list-of-object 'object)`
			`(tgapi-util-get-edge-zone-id-list-of-object "object")`

Feature		Description	Utility Function
		Return a list of cell zones by ID in the specified object	`(tgapi-util-get-cell-zone-id-list-of-object 'object)`
			`(tgapi-util-get-cell-zone-id-list-of-object "object")`
		Return a list of face zones in the specified face zone labels of the object specified	`(get-face-zones-of-labels 'object '(list-of-label-names))`
		Return a list of face zones by ID in the specified face zone labels of an object	`(tgapi-util-get-face-zone-id-list-of-labels 'object '(face-zone-label-list))`
			`(tgapi-util-get-face-zone-id-list-of-labels "object" '(face-zone-label-list))`
		Return a list of face zones shared by regions of specified types in the mesh object specified, where `region-type` is `fluid-fluid`, `solid-solid`, or `fluid-solid`	`(get-face-zones-shared-by-regions-of-type 'mesh-object 'region-type)`
		Return a list of face zones in the specified regions	`(get-face-zones-of-regions 'object '(list-of-region-names))`
	By cell shape	Return a list of prism cell zones	`(tgapi-util-get-prism-cell-zones '(zone-list))`
			`(tgapi-util-get-prism-cell-zones "zone-name-pattern*")`
		Return a list of tet cell zones	`(tgapi-util-get-tet-cell-zones '(zone-list))`
			`(tgapi-util-get-tet-cell-zones "zone-name-pattern*")`
	By connectivity	Return adjacent cell zones for given face zone	`(tgapi-util-get-adjacent-cell-zones zone-id)`
			`(tgapi-util-get-adjacent-cell-zones 'zone-name)`
			`(tgapi-util-get-adjacent-cell-zones "zone-name")`
		Return adjacent boundary face zones for given cell zones.	`(tgapi-util-get-adjacent-face-zones '(cell-zone-list))`
			`(tgapi-util-get-adjacent-face-zones "cell-zone-name-pattern*")`
		Return adjacent interior and boundary face zones for given cell zones.	`(tgapi-util-get-adjacent-interior-and-boundary-face-zones '(cell-zone-list))`

Feature		Description	Utility Function
			`(tgapi-util-get-adjacent-interior-and-boundary-face-zones "cell-zone-name-pattern*")`
		Return adjacent zones based on edge connectivity	`(tgapi-util-get-adjacent-zones-by-edge-connectivity '(zone-list))`
			`(tgapi-util-get-adjacent-zones-by-edge-connectivity "zone-name-pattern*")`
		Return adjacent zones based on node connectivity	`(tgapi-util-get-adjacent-zones-by-node-connectivity '(zone-list))`
			`(tgapi-util-get-adjacent-zones-by-node-connectivity "zone-name-pattern*")`
		Returns the number of faces and the boundary face zones that are shared with the specified cell zones.	`(tgapi-util-get-shared-boundary-zones 'cell-zone-name)`
			`(tgapi-util-get-shared-boundary-zones '(cell-zone-list))`
			`(tgapi-util-get-shared-boundary-zones "cell-zone-name-pattern*")`
		Returns interior face zones connected to given cell zones.	`(tgapi-util-get-interior-zones-connected-to-cell-zones '(cell-zone-list))`
			`(tgapi-util-get-interior-zones-connected-to-cell-zones "cell-zone-name-pattern*")`
	Mesh setup	Return a list of face zones with zone-specific prism settings applied	`(tgapi-util-get-face-zones-with-zone-specific-prisms-applied)`
		Return a list of face zones to which the specified prism controls apply	`(tgapi-util-get-face-zones-of-prism-controls "control-name")`
		Return the baffle zones based on the face zone list specified	`(tgapi-util-get-baffles '(face-zone-list))`
		Return the embedded baffle zones	`(tgapi-util-get-embedded-baffles)`
	Wrapped zones	Return a list of wrapped face zones	`(get-wrapped-zones)`

Feature		Description	Utility Function
	Unreferenced zones	Return a list of unreferenced zones	`(get-unreferenced-edge-zones)`
			`(get-unreferenced-face-zones)`
			`(get-unreferenced-cell-zones)`
		Return a list of unreferenced zones whose names contain the specified filter string	`(get-unreferenced-edge-zones-of-filter 'filter)`
			`(get-unreferenced-face-zones-of-filter 'filter)`
			`(get-unreferenced-cell-zones-of-filter 'filter)`
		Return a list of unreferenced zones by ID, whose names contain the specified pattern	`(tgapi-util-get-unreferenced-face-zone-id-list-of-pattern "pattern*")`
			`(tgapi-util-get-unreferenced-cell-zone-id-list-of-pattern "pattern*")`
			`(tgapi-util-get-unreferenced-edge-zone-id-list-of-pattern "pattern*")`
	Mesh statistics	Return cell zone with maximum volume for given list or pattern of cell zones	`(tgapi-util-get-maxsize-cell-zone-by-volume "pattern*")`
			`(tgapi-util-get-maxsize-cell-zone-by-volume '(zone-list))`
		Return cell zone with maximum count of elements for given list or pattern of cell zones	`(tgapi-util-get-maxsize-cell-zone-by-count "pattern*")`
			`(tgapi-util-get-maxsize-cell-zone-by-count '(zone-list))`
		Return face zone with minimum area for given list or pattern of face zones	`(tgapi-util-get-minsize-face-zone-by-area "pattern*")`
			`(tgapi-util-get-minsize-face-zone-by-area '(zone-list))`
		Return face zone with minimum count of elements for given list or pattern of face zones	`(tgapi-util-get-minsize-face-zone-by-count "pattern*")`

Feature		Description	Utility Function
			`(tgapi-util-get-minsize-face-zone-by-count '(zone-list))`
		Return a list of face zones with a count below the maximum entity count (`maximum-entity-count`) specified. You can choose to restrict the report to only boundary face zones, if required (`only-boundary?` set to `#t` or `#f`).	`(tgapi-util-get-face-zone-list-by-maximum-entity-count maximum-entity-count only-boundary?)`
		Return a list of edge zones with a count below the maximum entity count (`maximum-entity-count`) specified. You can choose to restrict the report to only boundary edge zones, if required (`only-boundary?` set to `#t` or `#f`).	`(tgapi-util-get-edge-zone-list-by-maximum-entity-count maximum-entity-count only-boundary?)`
		Return a list of cell zones with a count below the maximum entity count (`maximum-entity-count`) specified.	`(tgapi-util-get-cell-zone-list-by-maximum-entity-count maximum-entity-count)`
		Return a list of face zones with a maximum zone area below the `maximum-zone-area` specified.	`(tgapi-util-get-face-zone-list-by-maximum-zone-area maximum-zone-area)`
		Return a list of face zones with a minimum zone area above the `minimum-zone-area` specified.	`(tgapi-util-get-face-zone-list-by-minimum-zone-area minimum-zone-area)`
	Mesh diagnostics	Return a list of zones with free faces for the face zones specified	`(tgapi-util-get-zones-with-free-faces '(zone-id-list))`
			`(tgapi-util-get-zones-with-free-faces '(zone-name-list))`
			`(tgapi-util-get-zones-with-free-faces "zone-name-pattern*")`
		Return a list of zones with multi-connected faces for the face zones specified	`(tgapi-util-get-zones-with-multi-faces '(zone-id-list))`
			`(tgapi-util-get-zones-with-multi-faces '(zone-name-list))`
			`(tgapi-util-get-zones-with-multi-faces "zone-name-pattern*")`

Feature		Description	Utility Function
		Return a list of overlapping face zones based on the `area-tolerance` and `distance-tolerance` specified	`(tgapi-util-get-overlap-ping-face-zones "face-zone-name-pattern*" area-toler-ance distance-tolerance)`
		Return a list of zones with marked faces for the face zones specified	`(tgapi-util-get-zones-with-marked-faces '(zone-id-list))`
			`(tgapi-util-get-zones-with-marked-faces '(zone-name-list))`
			`(tgapi-util-get-zones-with-marked-faces "zone-name-pattern*")`
Object lists	By name	Return a list of all objects	`(tgapi-util-get-all-object-name-list)`
	By type	Return a list of objects of the specified type	`(get-objects-of-type 'type)`
			`(tgapi-util-get-object-name-list-of-type 'type)`
	By filter	Return a list of objects whose names contain the specified filter string	`(get-objects-of-filter 'filter)`
Region lists	By object	Return a list of regions in the specified object	`(get-regions-of-object 'ob-ject)`
			`(tgapi-util-get-region-name-list-of-object 'ob-ject)`
			`(tgapi-util-get-region-name-list-of-object "ob-ject")`
		Returns a sorted list of volumetric regions by volume for the object specified. Specify the `order` (ascend-ing or descending).	`(tgapi-util-sort-regions-by-volume "object-name" "order")`
		Return the region volume for the specified region of an object	`(tgapi-util-get-region-volume 'object-name 're-gion-name)`
			`(tgapi-util-get-region-volume "object-name" "re-gion-name")`
	By filter	Return a list of regions in the specified object, whose names contain the specified filter string	`(get-regions-of-filter 'ob-ject 'filter)`
	By name pattern	Return a list of regions in the specified object, whose names contain the specified name pattern	`(tgapi-util-get-region-name-list-of-pattern 'ob-`

Feature		Description	Utility Function
			`ject "region-name-pat-tern*")`
			`(tgapi-util-get-region-name-list-of-pattern "ob-ject" "region-name-pat-tern*")`
	By face zones	Return a list of regions containing the face zones specified	`(get-regions-of-face-zones '(list-of-face-zone-IDs))`
			`(tgapi-util-get-region-name-list-of-face-zones '(list-of-face-zone-IDs))`
			`(tgapi-util-get-region-name-list-of-face-zones "face-zone-name-pattern*")`
Overlapping zone pairs	Based on join tolerance and feature angle	Return the pairs of overlapping face zones based on the join tolerance and feature angle. • Specify the face zones using a `face-zone-name-list`, `face-zone-ID-list`, or `face-zone-name-pattern*`. • Specify the tolerance value for locating the overlapping faces (`join-tolerance`). • Choose to use an absolute tolerance value or relative to face edges (set `absolute-tolerance` to #t or #f). • Specify the feature angle to identify features in the overlap region (`feature-angle`). The default value is 40. Each member in the list returned includes the zone IDs for the overlapping zone pair and the join region represented by the bounding box. The same pair of zones may appear multiple times (with different join region bounding box coordinates) in the returned list. The returned list is of the format:	`(tgapi-util-find-join-pairs '(face-zone-name-list) join-tolerance absolute-tolerance? join-angle)`
			`(tgapi-util-find-join-pairs '(face-zone-id-list) join-tolerance absolute-toler-ance? join-angle)`
			`(tgapi-util-find-join-pairs "face-zone-name-pattern*" join-tolerance absolute-tolerance? join-angle)`

Feature		Description	Utility Function

		`((pair1_zone-id1 pair1_zone-id2 (join-region-bounding-box-min x y z) (join-region-bounding-box-max x y z)) (pair2_zone-id1 pair2_zone-id2 (join-region-bounding-box-min x y z) (join-region-bounding-box-max x y z)) ...)`
		
		For example:
		`((2 5 (-211.40 -139.24 -165.17) (46.36 -101.69 -124.61)) (2 5 (-211.4 -204.94 -55.89) (46.29 -171.68 -12.16)) (7 8 (-170.82 140.79 -108.57) (5.83 162.47 -89.04)))`	

List operations	Convert	Convert a list of zone name symbols to a list of IDs	`(tgapi-util-convert-zone-name-symbols-to-ids '(zone-name-list))`
		Convert a list of zone name strings to a list IDs	`(tgapi-util-convert-zone-name-strings-to-ids '(zone-name-list))`
		Convert a list of zone IDs to a list of name strings	`(tgapi-util-convert-zone-ids-to-name-strings '(zone-id-list))`
		Convert a list of zone IDs to a list of name symbols	`(tgapi-util-convert-zone-ids-to-name-symbols '(zone-id-list))`
		Convert a list of symbols to strings	`(tgapi-util-convert-symbol-list-to-string '(symbol-list))`
		Create a string from a list of symbols	`(tgapi-util-create-string-from-symbol-list '(symbol-list))`
	Subtract	Returns the boolean subtraction of the lists	`(tgapi-util-integer-list-subtract '(list-1) '(list-2))`
	Replace	Enables you to replace one element in a list by another	`(tgapi-util-list-replace new-element old-element input-list)`
	Remove	Enables you to remove an element from a list	`(tgapi-util-remove-element-from-list input-list element)`

Feature		Description	Utility Function
	Search	Enables you to search for an element in a list.	`(tgapi-util-list-contains? input-list element)`

A.1.1. Using Boolean Operations with Lists

You can perform Boolean operations on lists returned by the query functions:

- Union of lists: (list-union list-1 list-2 ...)

- Intersection of lists: (list-intersection list-1 list-2 ...)

- Subtraction of lists: (list-subtract list-1 list-2)

> **Note:**
>
> Only two lists can be used as arguments for the subtraction operation.

For example,

```
list-1 = '(1 2 3 4)
list-2 = '(1 5 6)
list-3 = '(1 4 7)
```

then:

```
(list-union list-1 list-2 list-3) = '(1 2 3 4 5 6 7)
(list-intersection list-1 list-2 list-3) = '(1)
(list-subtract list-1 list-2) = '(2 3 4)
```

A.1.2. Examples

- Smoothing the **prism-cap** zone created during the prism creation operation:

  – Obtain a list of the zones named **prism-cap***.

    ```
    (define initial-zones(get-face-zones-of-filter 'prism-cap*))
    ```

  – Apply appropriate prism parameters and create prisms.

  – Obtain a list of the zones named **prism-cap*** after the prism creation operation.

    ```
    (define final-zones(get-face-zones-of-filter 'prism-cap*))
    ```

  – Smooth the recently created **prism-cap** zone.

    ```
    /boundary/improve/smooth(eval-expr '(list-subtract final-zones ini-
    tial-zones))
    ```

- Improving mesh object quality:

```
/objects/improve-object-quality (get-objects-of-type 'mesh) smooth-
and-improve 1 no
```

- Recovering periodic surfaces for mesh object face zones:

```
/boundary/recover-periodic-surfaces auto (get-face-zones-of-label
'_fluid '(periodic-1))
```

- Defining curvature sizing for all object face zones except symmetry planes:

```
/scoped-sizing/create control-curv curvature object-faces yes no
"(eval-expr '(list-subtract (get-objects-of-filter '*) (get-objects-
of-filter 'sym*)))" 0.25 4 1.2 18
```

A.2. Label Utility Functions

The following label utility functions are available:

Feature		Description	Function
Add labels	On face zones	Add labels on the specified face zones. • Specify the face zones using a `face-zone-name-list`, `face-zone-ID-list`, or `face-zone-name-pattern*`. • Specify the labels using a `label-name-list` (for example `'(label1 label2)`).	`(tgapi-util-add-labels-on-face-zones '(face-zone-name-list) '(label-name-list))`
			`(tgapi-util-add-labels-on-face-zones '(face-zone-ID-list) '(label-name-list))`
			`(tgapi-util-add-labels-on-face-zones "face-zone-name-pattern*" '(label-name-list))`
	On cell zones	Add labels on the specified cell zones. • Specify the cell zones using a `cell-zone-name-list`, `cell-zone-ID-list`, or `cell-zone-name-pattern*`. • Specify the labels using a `label-name-list` (for example `'(label1 label2)`).	`(tgapi-util-add-labels-on-cell-zones '(cell-zone-name-list) '(label-name-list))`
			`(tgapi-util-add-labels-on-cell-zones '(cell-zone-ID-list) '(label-name-list))`
			`(tgapi-util-add-labels-on-cell-zones "cell-zone-name-pattern*" '(label-name-list))`
	On edge zones	Add labels on the specified edge zones. • Specify the edge zones using an `edge-zone-name-list`, `edge-zone-ID-list`, or `edge-zone-name-pattern*`. • Specify the labels using a `label-name-list` (for example `'(label1 label2)`).	`(tgapi-util-add-labels-on-edge-zones '(edge-zone-name-list) '(label-name-list))`
			`(tgapi-util-add-labels-on-edge-zones '(edge-zone-ID-list) '(label-name-list))`
			`(tgapi-util-add-labels-on-edge-zones "edge-zone-name-pattern*" '(label-name-list))`

Feature		Description	Function
Return list of labels	On face zones	Returns the list of labels for the specified face zones. Specify the face zones using a `face-zone-name-list`, `face-zone-ID-list`, or `face-zone-name-pattern*`.	`(tgapi-util-get-labels-on-face-zones '(face-zone-name-list))`
			`(tgapi-util-get-labels-on-face-zones '(face-zone-ID-list))`
			`(tgapi-util-get-labels-on-face-zones "face-zone-name-pattern*")`
	On cell zone	Returns the list of labels for the specified cell zones. Specify the cell zones using a `cell-zone-name-list`, `cell-zone-ID-list`, or `cell-zone-name-pattern*`.	`(tgapi-util-get-labels-on-cell-zones '(cell-zone-name-list))`
			`(tgapi-util-get-labels-on-cell-zones '(cell-zone-ID-list))`
			`(tgapi-util-get-labels-on-cell-zones "cell-zone-name-pattern*")`
	On edge zone	Returns the list of labels for the specified edge zones. Specify the edge zones using an `edge-zone-name-list`, `edge-zone-ID-list`, or `edge-zone-name-pattern*`.	`(tgapi-util-get-labels-on-edge-zones '(edge-zone-name-list))`
			`(tgapi-util-get-labels-on-edge-zones '(edge-zone-ID-list))`
			`(tgapi-util-get-labels-on-edge-zones "edge-zone-name-pattern*")`
	Face Zone Label lists by object	Return a list of face zone labels in the specified object.	`(get-labels-of-object 'object)`
			`(get-labels-of-object "object")`
			`(tgapi-util-get-all-label-name-list 'object)`
			`(tgapi-util-get-all-label-name-list "object")`
		Return a list of face zone labels in the specified object, whose names contain the specified filter/pattern string.	`(get-labels-of-filter 'object 'filter)`
			`(tgapi-util-get-label-name-list-of-pattern 'object "label-name-pattern*")`
			`(tgapi-util-get-label-name-list-of-pattern "object" "label-name-pattern*")`
	Face Zone Label lists by face zones	Return a list of face zone labels containing the face zones specified.	`(get-labels-of-face-zones '(list-of-face-zone-IDs))`

Feature		Description	Function
Return list of zones with labels	Face zones	Returns the list of face zones (by ID) containing the labels specified. • Specify the face zones using a `face-zone-name-list`, `face-zone-ID-list`, or `face-zone-name-pattern*`. • Specify the labels using a `label-name-list` (for example `'(label1 label2)`).	`(tgapi-util-get-face-zone-id-list-with-labels '(face-zone-name-list) '(label-name-list))`
			`(tgapi-util-get-face-zone-id-list-with-labels '(face-zone-ID-list) '(label-name-list))`
			`(tgapi-util-get-face-zone-id-list-with-labels "face-zone-name-pattern* '(label-name-list))`
	Cell Zones	Returns the list of cell zones (by ID) containing the labels specified. • Specify the cell zones using a `cell-zone-name-list`, `cell-zone-ID-list`, or `cell-zone-name-pattern*`. • Specify the labels using a `label-name-list` (for example `'(label1 label2)`).	`(tgapi-util-get-cell-zone-id-list-with-labels '(cell-zone-name-list) '(label-name-list))`
			`(tgapi-util-get-cell-zone-id-list-with-labels '(cell-zone-ID-list) '(label-name-list))`
			`(tgapi-util-get-cell-zone-id-list-with-labels "cell-zone-name-pattern* '(label-name-list))`
	Edge zones	Returns the list of edge zones (by ID) containing the labels specified. • Specify the face zones using an `edge-zone-name-list`, `edge-zone-ID-list`, or `edge-zone-name-pattern*`. • Specify the labels using a `label-name-list` (for example `'(label1 label2)`).	`(tgapi-util-get-edge-zone-id-list-with-labels '(edge-zone-name-list) '(label-name-list))`
			`(tgapi-util-get-edge-zone-id-list-with-labels '(edge-zone-ID-list) '(label-name-list))`
			`(tgapi-util-get-edge-zone-id-list-with-labels "edge-zone-name-pattern* '(label-name-list))`
Remove labels from zones	From face zones	Removes the specified labels from the face zones. • Specify the face zones using a `face-zone-name-list`, `face-zone-ID-list`, or `face-zone-name-pattern*`. • Specify the labels using a `label-name-list` (for example `'(label1 label2)`).	`(tgapi-util-remove-labels-on-face-zones '(face-zone-name-list) '(label-name-list))`
			`(tgapi-util-remove-labels-on-face-zones '(face-zone-ID-list) '(label-name-list))`
			`(tgapi-util-remove-labels-on-face-zones "face-zone-name-pattern*" '(label-name-list))`

Feature		Description	Function
	From cell zones	Removes the specified labels from the cell zones. • Specify the cell zones using a `cell-zone-name-list`, `cell-zone-ID-list`, or `cell-zone-name-pattern*`. • Specify the labels using a `label-name-list` (for example `'(label1 label2)`).	`(tgapi-util-remove-labels-on-cell-zones '(cell-zone-name-list) '(label-name-list))`
			`(tgapi-util-remove-labels-on-cell-zones '(cell-zone-ID-list) '(label-name-list))`
			`(tgapi-util-remove-labels-on-cell-zones "cell-zone-name-pattern*" '(label-name-list))`
	From edge zones	Removes the specified labels from the edge zones. • Specify the edge zones using an `edge-zone-name-list`, `edge-zone-ID-list`, or `edge-zone-name-pattern*`. • Specify the labels using a `label-name-list` (for example `'(label1 label2)`).	`(tgapi-util-remove-labels-on-edge-zones '(edge-zone-name-list) '(label-name-list))`
			`(tgapi-util-remove-labels-on-edge-zones '(edge-zone-ID-list) '(label-name-list))`
			`(tgapi-util-remove-labels-on-edge-zones "edge-zone-name-pattern*" '(label-name-list))`

A.2.1. Examples

Consider a model with face zones **part1**, **part2**, **part3**, **inlet1**, **inlet2**, and **outlet**.

• Add labels on the specified face zones.

```
(tgapi-util-add-labels-on-face-zones '(inlet1 inlet2) '(inlet))

(tgapi-util-add-labels-on-face-zones "part*" '(wall solid))
```

• Return the list of labels on the specified face zone.

```
(tgapi-util-get-labels-on-face-zones '(12 34))
```

Returns list of labels: `(inlet)`

```
(tgapi-util-get-labels-on-face-zones "part*")
```

Returns list of labels: `(wall solid)`

• Remove labels from the specified face zones.

```
(tgapi-util-remove-labels-on-face-zones "part*" '(solid))
```

• Return the list of face zones (by ID) containing the labels specified.

```
(tgapi-util-get-face-zone-id-list-with-labels "inlet*" '(inlet))
```

Returns face zone ID list: `(12 34)`

A.3. Report Utility Functions

The following functions are available for reporting mesh quality, statistics, face/cell distribution:

Feature	Description	Utility Function
Report Mesh Types	Return zone type as integer	`(tgapi-util-get-zone-type zone-id)`
		`(tgapi-util-get-zone-type zone-name)`
	Return cell zone shape as symbol	`(tgapi-util-get-cell-zone-shape zone-id)`
Report Mesh Statistics	Return count of entities for face zones	`(tgapi-util-get-face-zone-count '(zone-id-list))`
		`(tgapi-util-get-face-zone-count '(zone-name-list))`
		`(tgapi-util-get-face-zone-count "zone-name-pattern*")`
	Return count of entities for cell zones	`(tgapi-util-get-cell-zone-count '(zone-id-list))`
		`(tgapi-util-get-cell-zone-count '(zone-name-list))`
		`(tgapi-util-get-cell-zone-count "zone-name-pattern*")`
	Returns the node count for the specified face zone.	`(tgapi-util-get-face-zone-node-count 'zone-id)`
		`(tgapi-util-get-face-zone-node-count 'zone-name)`
		`(tgapi-util-get-face-zone-node-count "zone-name")`
	Return face zone area for the specified zones	`(tgapi-util-get-face-zone-area '(zone-id-list))`
		`(tgapi-util-get-face-zone-area '(zone-name-list))`
		`(tgapi-util-get-face-zone-area "zone-name-pattern*")`
	Return cell zone volume for the specified zones	`(tgapi-util-get-cell-zone-volume '(zone-id-list))`
		`(tgapi-util-get-cell-zone-volume '(zone-name-list))`
		`(tgapi-util-get-cell-zone-volume "zone-name-pattern*")`

Feature	Description	Utility Function

Report Mesh Distribution	Report the face mesh distribution based on the specified `measure`, `partitions`, and `range` (such as `(0.9 1)`).
	
	Specify the `"measure"` as one of the following:
	
	• `"Skewness"`
	
	• `"Equiangle Skewness"`
	
	• `"Size Change"`
	
	• `"Edge Ratio"`
	
	• `"Size"`
	
	• `"Aspect Ratio"`
	
	• `"Squish"`
	
	• `"Warp"`
	
	• `"Dihedral Angle"`
	
	• `"ICEMCFD Quality"`
	
	• `"Ortho Skew"`
	
	• `"FLUENT Aspect Ratio"`
	
	• `"Inverse Orthogonal Quality"`
		`(tgapi-util-get-face-mesh-distribution '(face-zone-list) "measure" partitions '(range))`
		
		`(tgapi-util-get-face-mesh-distribution "face-zone-name-pattern*" "measure" partitions '(range))`

	Report the cell mesh distribution based on the specified `measure`, `partitions`, and `range` (such as `(0.9 1)`).
	
	Specify the `"measure"` as one of the following:
	
	• `"Skewness"`
	
	• `"Equiangle Skewness"`
	
	• `"Size Change"`
	
	• `"Edge Ratio"`
	
	• `"Size"`
	
	• `"Aspect Ratio"`
		`(tgapi-util-get-cell-mesh-distribution '(cell-zone-list) "measure" partitions '(range))`
		
		`(tgapi-util-get-cell-mesh-distribution "cell-zone-name-pattern*" "measure" partitions '(range))`

Feature	Description	Utility Function
	• `"Squish"` • `"Warp"` • `"Dihedral Angle"` • `"ICEMCFD Quality"` • `"Ortho Skew"` • `"FLUENT Aspect Ratio"` • `"Inverse Orthogonal Quality"`	

Report Quality/Size Limits	Report the worst quality face and face quality limits (minimum, maximum, average quality) for the list of zones based on the `measure` specified. You can also report face size limits.
	
	 Specify the `"measure"` as one of the following:
	
	 • `"Skewness"`
	
	 • `"Equiangle Skewness"`
	
	 • `"Size Change"`
	
	 • `"Edge Ratio"`
	
	 • `"Size"`
	
	 • `"Aspect Ratio"`
	
	 • `"Squish"`
	
	 • `"Warp"`
	
	 • `"Dihedral Angle"`
	
	 • `"ICEMCFD Quality"`
	
	 • `"Ortho Skew"`
	
	 • `"FLUENT Aspect Ratio"`
	
	 • `"Inverse Orthogonal Quality"`
		`(tgapi-util-get-face-quality-limits '(face-zone-list) "measure")`
		
		 `(tgapi-util-get-face-quality-limits "face-zone-name-pattern*" "measure")`

Feature	Description	Utility Function
	Report the number of cells and the cell quality limits (minimum, maximum, average quality) for the list of zones based on the `measure` specified. You can also report the cell size limits. Specify the `"measure"` as one of the following: • `"Skewness"` • `"Equiangle Skewness"` • `"Size Change"` • `"Edge Ratio"` • `"Size"` • `"Aspect Ratio"` • `"Squish"` • `"Warp"` • `"Dihedral Angle"` • `"ICEMCFD Quality"` • `"Ortho Skew"` • `"FLUENT Aspect Ratio"` • `"Inverse Orthogonal Quality"`	`(tgapi-util-get-cell-quality-limits '(cell-zone-list) "measure")`
		`(tgapi-util-get-cell-quality-limits "cell-zone-name-pat-tern*" "measure")`
	Report the worst quality cell (ID and location) for the cell zones based on the `measure` specified. Specify the `"measure"` as one of the following: • `"Skewness"` • `"Equiangle Skewness"` • `"Size Change"` • `"Edge Ratio"` • `"Size"`	`(tgapi-util-print-worst-qual-ity-cell '(cell-zone-list) "measure")`
		`(tgapi-util-print-worst-qual-ity-cell "cell-zone-name-pat-tern*" "measure")`

Feature	Description	Utility Function
	• "Aspect Ratio" • "Squish" • "Warp" • "Dihedral Angle" • "ICEMCFD Quality" • "Ortho Skew" • "FLUENT Aspect Ratio" • "Inverse Orthogonal Quality"	
	Report the edge size limits for the list of face zones	`(tgapi-util-get-edge-size-limits '(face-zone-list))`
		`(tgapi-util-get-edge-size-limits "face-zone-name-pat-tern*")`
Report count	Returns the count of free faces for the face zones specified	`(tgapi-util-get-free-faces-count '(face-zone-name-list))`
		`(tgapi-util-get-free-faces-count '(face-zone-id-list))`
		`(tgapi-util-get-free-faces-count "face-zone-name-pat-tern*")`
	Returns the count of multi-connected faces for the face zones specified	`(tgapi-util-get-multi-faces-count '(face-zone-name-list))`
		`(tgapi-util-get-multi-faces-count '(face-zone-id-list))`
		`(tgapi-util-get-multi-faces-count "face-zone-name-pat-tern*")`
	Returns the count of marked faces for the face zones specified	`(tgapi-util-count-marked-faces '(face-zone-name-list))`
		`(tgapi-util-count-marked-faces '(face-zone-id-list))`
		`(tgapi-util-count-marked-faces "face-zone-name-pat-tern*")`
Print summary	Print the diagnostics summary for a list of objects	`(tgapi-util-print-dia-gnostics-summary '(object-name-list))`

Feature	Description	Utility Function
Mesh check	Reports the selected mesh check statistics for the zones specified. • Specify the `"type-name"` as one of the following: – `"bounding-box-statistics"` – `"volume-statistics"` – `"face-area-statistics"` – `"nodes-per-edge"` – `"nodes-per-face"` – `"nodes-per-cell"` – `"faces-or-neighbors-per-cell"` – `"cell-faces-or-neighbors"` – `"isolated-cells"` – `"face-handedness"` – `"periodic-face-pairs"` – `"face-children"` – `"zone-boundary-conditions"` – `"invalid-node-coordinates"` – `"poly-cells"` – `"parallel-invalid-zones"` – `"parallel-invalid-neighborhood"` – `"parallel-invalid-interfaces"` Refer to Checking the Mesh in the *Fluent User's Guide* for	`(tgapi-util-mesh-check "type-name" '(edge-zone-list) '(face-zone-list) '(cell-zone-list))`
		`(tgapi-util-mesh-check "type-name" "edge-zone-name-pattern*" '(face-zone-list) '(cell-zone-list))`
		`(tgapi-util-mesh-check "type-name" '(edge-zone-list) "face-zone-name-pattern*" '(cell-zone-list))`
		`(tgapi-util-mesh-check "type-name" "edge-zone-name-pattern*" "face-zone-name-pattern*" '(cell-zone-list))`
		`(tgapi-util-mesh-check "type-name" '(edge-zone-list) '(face-zone-list) "cell-zone-name-pattern*")`
		`(tgapi-util-mesh-check "type-name" "edge-zone-name-pattern*" "face-zone-name-pattern*" "cell-zone-name-pattern*")`

Feature	Description	Utility Function
	descriptions of the individual mesh checks. • Specify the zones to be checked. Specify the edge zones (`edge-zone-list` or `edge-zone-name-pattern*`), face zones (`face-zone-list` or `face-zone-name-pattern*`), and cell zones (`cell-zone-list` or `cell-zone-name-pattern*`).	

A.4. Diagnostic Based Marking Utility Functions

The following functions are available for identifying faces/zones based on diagnostic criteria such as face connectivity and quality:

Feature Description	Utility Function
Mark free faces on the face zones specified	`(tgapi-util-mark-free-faces '(zone-id-list))`
	`(tgapi-util-mark-free-faces '(zone-name-list))`
	`(tgapi-util-mark-free-faces "zone-name-pattern*")`
Mark multi-connected faces on the face zones specified based on fringe length (n)	`(tgapi-util-mark-multi-faces '(zone-id-list) n)`
	`(tgapi-util-mark-multi-faces '(zone-name-list) n)`
	`(tgapi-util-mark-multi-faces "zone-name-pattern*" n)`
Mark self-intersecting faces on the face zones specified. Specify whether to mark folded faces or not (`mark-folded?` set to `#t` or `#f`).	`(tgapi-util-mark-self-intersecting-faces '(zone-id-list) mark-folded?)`
	`(tgapi-util-mark-self-intersecting-faces '(zone-name-list) mark-folded?)`
	`(tgapi-util-mark-self-intersecting-faces "zone-name-pattern*" mark-folded?)`
Mark faces in self-proximity on the face zones specified. Specify whether to use relative tolerance (`relative-tolerance?` set to `#t` or `#f`), tolerance value, the `angle`, and whether to ignore orientation (`ignore-orientation?` set to `#t` or `#f`).	`(tgapi-util-mark-faces-in-self-proximity '(zone-id-list) relative-tolerance? tolerance angle ignore-orientation?)`
	`(tgapi-util-mark-faces-in-self-proximity '(zone-name-list) relative-`

Feature Description	Utility Function
	`tolerance? tolerance angle ignore-orientation?)`
	`(tgapi-util-mark-faces-in-self-prox-imity "zone-name-pattern*" relative-tolerance? tolerance angle ignore-orientation?)`
Mark duplicate faces on the face zones specified	`(tgapi-util-mark-duplicate-faces '(zone-id-list))`
	`(tgapi-util-mark-duplicate-faces '(zone-name-list))`
	`(tgapi-util-mark-duplicate-faces "zone-name-pattern*")`
Mark invalid normal locations on the face zones specified	`(tgapi-util-mark-invalid-normals '(zone-id-list))`
	`(tgapi-util-mark-invalid-normals '(zone-name-list))`
	`(tgapi-util-mark-invalid-normals "zone-name-pattern*")`
Mark point contact locations on the face zones specified	`(tgapi-util-mark-point-contacts '(zone-id-list))`
	`(tgapi-util-mark-point-contacts '(zone-name-list))`
	`(tgapi-util-mark-point-contacts "zone-name-pattern*")`
Mark island faces on the face zones specified, based on the island face count (n)	`(tgapi-util-mark-island-faces '(zone-id-list) n)`
	`(tgapi-util-mark-island-faces '(zone-name-list) n)`
	`(tgapi-util-mark-island-faces "zone-name-pattern*" n)`
Mark spikes on the face zones specified, based on the spike angle (spike-angle)	`(tgapi-util-mark-spikes '(zone-id-list) spike-angle)`
	`(tgapi-util-mark-spikes '(zone-name-list) spike-angle)`
	`(tgapi-util-mark-spikes "zone-name-pattern*" spike-angle)`
Mark steps on the face zones specified, based on the step angle (step-angle) and step width (step-width)	`(tgapi-util-mark-steps '(zone-id-list) step-angle step-width)`
	`(tgapi-util-mark-steps '(zone-name-list) step-angle step-width)`
	`(tgapi-util-mark-steps "zone-name-pattern*" step-angle step-width)`

Feature Description	Utility Function
Mark sliver faces on the face zones specified, based on the maximum height (`max-height`) and skewness limit (`skew-limit`)	`(tgapi-util-mark-sliver-faces '(zone-id-list) max-height skew-limit)`
	`(tgapi-util-mark-sliver-faces '(zone-name-list) max-height skew-limit)`
	`(tgapi-util-mark-sliver-faces "zone-name-pattern*" max-height skew-limit)`
Mark bad quality faces on the boundary face zones specified, based on the quality limit (`quality-limit`) and number of rings (`nrings`).	`(tgapi-util-mark-bad-quality-faces '(zone-id-list) quality-limit nrings)`
	`(tgapi-util-mark-bad-quality-faces '(zone-name-list) quality-limit nrings)`
	`(tgapi-util-mark-bad-quality-faces "zone-name-pattern*" quality-limit nrings)`
Mark faces based on the `quality-measure` and `quality-limit` specified. Specify whether to append the faces to those previously marked or clear previously marked faces (`append-marking?` set to `#t` or `#f`)	`(tgapi-util-mark-faces-by-quality '(zone-id-list) quality-measure quality-limit append-marking?)`
	`(tgapi-util-mark-faces-by-quality '(zone-name-list) quality-measure quality-limit append-marking?)`
	`(tgapi-util-mark-faces-by-quality "zone-name-pattern*" quality-measure quality-limit append-marking?)`
Mark face strips based on the `strip-type`, `strip-height`, `quality-measure`, `quality-limit`, and `feature-angle` specified. Possible values for `strip-type` are: • 1: boundary-boundary strip. Multi-connected face edges are also considered as boundary here. • 2: feature-feature strip between angle based features. Feature edges, multi-connected edges, and free edges are angle based features. Boundary edges will be considered features if there is an angle. • 3: all-all strip between all boundaries and features. • 4: pure feature-feature strip. Only pure features, boundary edges and multi edges will not be considered as pure feature edges even if there is an angle based feature. The recommended value is 2.	`(tgapi-util-mark-face-strips-by-height-and-quality '(zone-id-list) strip-type strip-height quality-measure quality-limit feature-angle)`
	`(tgapi-util-mark-face-strips-by-height-and-quality '(zone-name-list) strip-type strip-height quality-measure quality-limit feature-angle)`
	`(tgapi-util-mark-face-strips-by-height-and-quality "zone-name-pattern*" strip-type strip-height quality-measure quality-limit feature-angle)`

Feature Description	Utility Function
Mark all faces at nodes based on deviation from the size field. • Specify the zones using `zone-id-list`, `zone-name-list`, or `zone-name-pattern*`. • Specify the `min-size-factor` and `max-size-factor`. • Specify the size field type to be used to get size at node. Set `"sf-type-to-compare"` to `"volumetric"` or `"geodesic"`. Faces will be marked if the minimum edge length at the node is less than `min-size-factor×size-at-node` or the maximum edge length is greater than `max-size-factor×size-at-node`.	`(tgapi-util-mark-faces-deviating-from-size-field '(zone-id-list) min-size-factor max-size-factor "sf-type-to-compare")`
	`(tgapi-util-mark-faces-deviating-from-size-field '(zone-name-list) min-size-factor max-size-factor "sf-type-to-compare")`
	`(tgapi-util-mark-faces-deviating-from-size-field "zone-name-pattern*" min-size-factor max-size-factor "sf-type-to-compare")`
Mark all faces with node degree above the specified threshold. `Node degree` is defined as the number of edges connected to the node. • Specify the face zones using `zone-id-list`, `zone-name-list`, or `zone-name-pattern*`. • Specify the `node-degree-threshold`.	`(tgapi-util-mark-faces-using-node-degree '(zone-id-list) node-degree-threshold)`
	`(tgapi-util-mark-faces-using-node-degree '(zone-name-list) node-degree-threshold)`
	`(tgapi-util-mark-faces-using-node-degree "zone-name-pattern*" node-degree-threshold)`

A.5. Mesh Setup Utility Functions

The following functions are available for mesh setup:

Feature Description	Utility Function
Return a suitable average point based on the zones specified	`(get-average-bounding-box-center '(face-zone-ID-list))`
Return the bounding box extents for the list of zones.	`(tgapi-util-get-bounding-box-of-zone-list '(zone-id-list))`
Enables you to unpreserve some/all preserved cell zones during the meshing process.	`(tgapi-util-un-preserve-cell-zones '(cell-zone-list))`
	`(tgapi-util-un-preserve-cell-zones "cell-zone-name-pattern*")`
Create bodies of influence and if required body of influence size functions from the mesh refinement regions. Specify the refinement region type (set `"region-type"` to `"tet"` or `"hexcore"`).	`(tgapi-util-create-boi-and-size-functions-from-refinement-regions "region-type" "boi-prefix-string" create-size-function?)`

Feature Description	Utility Function
Specify the prefix for the BOI zones (`"boi-pre-fix-string"`), and choose whether to create the size functions (set `create-size-function?` to `#t` or `#f`).	
Enables you to scale the face zones around a pivot point or the bounding box center. Specify the face zones, the scale factors in the X, Y, Z directions (`scale`), the pivot point (`pivot`), and choose whether to use the bounding box center (`use-bbox-center?` set to `#t` or `#f`).	`(tgapi-util-scale-face-zones-around-pivot '(face-zone-list) '(scale) '(pivot) use-bbox-center?)`
	`(tgapi-util-scale-face-zones-around-pivot "face-zone-name-pattern*" '(scale) '(pivot) use-bbox-center?)`
Enables you to scale the cell zones around a pivot point or the bounding box center. Specify the cell zones, the scale factors in the X, Y, Z directions (`scale`), the pivot point (`pivot`), and choose whether to use the bounding box center (`use-bbox-center?` set to `#t` or `#f`).	`(tgapi-util-scale-cell-zones-around-pivot '(cell-zone-list) '(scale) '(pivot) use-bbox-center?)`
	`(tgapi-util-scale-cell-zones-around-pivot "cell-zone-name-pattern*" '(scale) '(pivot) use-bbox-center?)`
Return the face zones and their orientation for the mesh file specified	`(tgapi-util-dump-face-zone-orienta-tion-in-region "filename")`
Set the quality measure. Specify the `"measure"` as one of the following: • `"Orthogonal Quality"` • `"Skewness"` • `"Equiangle Skewness"` • `"Size Change"` • `"Edge Ratio"` • `"Size"` • `"Aspect Ratio"` • `"Squish"` • `"Warp"` • `"Dihedral Angle"` • `"ICEMCFD Quality"` • `"Ortho Skew"` • `"FLUENT Aspect Ratio"` • `"Inverse Orthogonal Quality"`	`(tgapi-util-set-quality-measure "measure")`

Feature Description	Utility Function
Set object cell zone type	`(tgapi-util-set-object-cell-zone-type 'object 'cell-zone-type)`
	`(tgapi-util-set-object-cell-zone-type "object" 'cell-zone-type)`
Set the number of compute threads to use for algorithms like mesh check and quality computation. You can use a variable number of compute threads for these algorithms depending on the current machine loads. The number of compute threads is between 2 and the value (`maximum-cores-available` - 1).	`(tgapi-util-set-number-of-parallel-compute-threads nthreads)`

A.6. Mesh Operation Utility Functions

Feature	Description	Utility Function
Rename zones	Renames an existing face zone	`(tgapi-util-rename-face-zone "zone-name" "new-name")`
		`(tgapi-util-rename-face-zone 'zone-id "new-name")`
	Renames an existing edge zone	`(tgapi-util-rename-edge-zone "zone-name" "new-name")`
		`(tgapi-util-rename-edge-zone 'zone-id "new-name")`
	Replace the face zone suffix to rename face zones. Specify whether to merge the face zones being renamed (set `merge?` to #t or #f). **Note:** If an empty string is specified for the `separator` (" "), the string specified for `replace-with` will be appended to the face zone names.	`(tgapi-util-replace-face-zone-suffix '(face-zone-list) "separator" "replace-with" merge?)`
	Replace the cell zone suffix to rename cell zones. Specify whether to merge the cell zones being renamed (set `merge?` to #t or #f)	`(tgapi-util-replace-cell-zone-suffix '(cell-zone-list) "old-suffix" "new-suffix" merge?)`
	Replace the edge zone suffix to rename edge zones. Specify whether to merge the edge zones being renamed (set `merge?` to #t or #f)	`(tgapi-util-replace-edge-zone-suffix '(edge-zone-list)`

Feature	Description	Utility Function
		`"old-suffix" "new-suffix" merge?)`
	Removes the ID suffix from face zone names	`(tgapi-util-remove-id-suffix-from-face-zones)`
	Clean up face zone names by removing IDs wherever possible	`(tgapi-util-clean-face-zone-names)`
	Remove the zone ID (`:id`) from zone ID list.	`(tgapi-util-remove-ids-from-zone-names '(zone-id-list))`
Change zone ID	Renumber zone IDs starting from the number specified (`start-number`).	`(tgapi-util-renumber-zone-ids '(zone-id-list) start-number)`
Rename objects	Renames the object	`(tgapi-util-rename-object "old-object-name" "new-object-name")`
	Rename objects by replacing the object suffix with a new suffix	`(tgapi-util-replace-object-suffix '(object-name) "separator" "new-suffix")`
Rename labels	Renames the face zone label	`(tgapi-util-rename-label "object-name" "old-label-name" "new-label-name")`
	Rename labels by replacing the label suffix with a new suffix	`(tgapi-util-replace-label-suffix '(object-list) "separator" "new-suffix")`
Copy labels	Copy labels from one face zone to another. Specify either face zone names or IDs	`(tgapi-util-copy-labels from-face-zone to-face-zone)`
Merge zones	Merges the specified face zones. Specify a list of zone IDs or name pattern.	`(tgapi-util-merge-face-zones '(zone-id-list))`
		`(tgapi-util-merge-face-zones "zone-name-pattern*")`
	Merges face zones of a given type based on name pattern	`(tgapi-util-merge-face-zone-of-type 'zone-type "zone-name-pattern*")`
	Merges face zones containing the specified prefix	`(tgapi-util-merge-face-zones-with-same-prefix "prefix")`
	Merges the specified cell zones. Specify a list of cell zones or name pattern.	`(tgapi-util-merge-cell-zones '(cell-zone-list))`
		`(tgapi-util-merge-cell-zones "cell-zone-name-pattern*")`
	Merge cell zones containing the specified prefix	`(tgapi-util-merge-cell-zones-with-same-prefix "prefix")`
	Merge cell zones containing the specified suffix	`(tgapi-util-merge-cell-zones-with-same-suffix "suffix")`

Feature	Description	Utility Function
Separate	Separates cells that are connected to specified face zones into another cell zone. This separation method applies only to prism cells. Specify the number of layers of cells (`nlayers`) to be separated.	`(tgapi-util-separate-cell-zone-layers-by-face-zone '(cell-zone-id) '(face-zone-list) nlayers)`
		`(tgapi-util-separate-cell-zone-layers-by-face-zone '(cell-zone-id) "face-zone-name-pattern*" nlayers)`
		`(tgapi-util-separate-cell-zone-layers-by-face-zone "cell-zone-name" '(face-zone-list) nlayers)`
		`(tgapi-util-separate-cell-zone-layers-by-face-zone "cell-zone-name" "face-zone-name-pattern*" nlayers)`
	Separate face zones based on the cell neighbors.	`(tgapi-util-separate-face-zones-by-cell-neighbor '(face-zone-list))`
		`(tgapi-util-separate-face-zones-by-cell-neighbor "face-zone-name-pattern*")`
Refine	Refine marked faces	`(tgapi-util-refine-marked-faces-in-zones '(face-zone-list))`
		`(tgapi-util-refine-marked-faces-in-zones "face-zone-name-pattern*")`
Patch	Fill holes associated with free edges for the face zones specified, based on the number of free edges (`max-hole-edges`)	`(tgapi-util-fill-holes-in-face-zone-list '(face-zone-list) max-hole-edges)`
		`(tgapi-util-fill-holes-in-face-zone-list "face-zone-name-pattern*" max-hole-edges)`
Project	Project a zone on the plane specified. Specify three points for defining the plane.	`(tgapi-util-project-zone-on-plane zone-id plane)`
Delete	Deletes all sub-domains (all domains other than **global**)	`(tgapi-util-delete-all-sub-domains)`
	Delete marked faces	`(tgapi-util-delete-marked-faces-in-zones '(face-zone-list))`

Feature	Description	Utility Function
		`(tgapi-util-delete-marked-faces-in-zones "face-zone-name-pattern*")`
	Delete empty zones based on the zones specified	`(tgapi-util-delete-empty-face-zones '(face-zone-list))`
		`(tgapi-util-delete-empty-face-zones "face-zone-name-pattern*")`
		`(tgapi-util-delete-empty-edge-zones '(edge-zone-list))`
		`(tgapi-util-delete-empty-edge-zones "edge-zone-name-pattern*")`
		`(tgapi-util-delete-empty-cell-zones '(cell-zone-list))`
		`(tgapi-util-delete-empty-cell-zones "cell-zone-name-pattern*")`
		`(tgapi-util-delete-empty-zones '(zone-list))`
		`(tgapi-util-delete-empty-zones "zone-name-pattern*")`
Validate mesh	Report if the boundary face zone exists	`(tgapi-util-boundary-zone-exists? zone-id)`
		`(tgapi-util-boundary-zone-exists? 'zone-name)`
	Report if the interior face zone exists	`(tgapi-util-interior-zone-exists? zone-id)`
		`(tgapi-util-interior-zone-exists? 'zone-name)`
	Report if the cell zone exists	`(tgapi-util-cell-zone-exists? zone-id)`
		`(tgapi-util-cell-zone-exists? 'zone-name)`
	Report if the volume mesh exists	`(tgapi-util-mesh-exists?)`

A.7. Miscellaneous Functions

Feature	Description	Utility Function
String operations	Check if the string contains another string	`(tgapi-util-string-contains? "string1*" "string2*")`

Feature	Description	Utility Function
	Enables you to split a string by the specified character.	`(tgapi-util-string-split "original-string" "split-character")`
	Enables you to split a string by the specified string.	`(tgapi-util-string-split-with-string "original-string" "split-string")`
	Enables you to replace a part (`old-sub-string`) of a `target-string` by the specified string (`new-sub-string`).	`(tgapi-util-string-replace "target-string" "old-sub-string" "new-sub-string")` For example: `(tgapi-util-string-replace "tst-string" "tst" "test")` will return `"test-string"`.
Folder/directory Management	Returns the path for the working directory	`(tgapi-util-get-mesher-working-directory)`
	Sync the Cortex current working directory with the Fluent current working directory (`directory`)	`(tgapi-util-sync-directories directory)`
File management	Finds the file in the working directory based on the `filename` and `file-suffix` specified.	`(tgapi-util-find-file-with-suffix "filename" "file-suffix")`
		`(tgapi-util-find-file-pattern-with-suffix "filename" "file-suffix")`
	Returns the last read file	`(tgapi-util-get-last-read-file-name)`
	Returns the file suffix for the last read file	`(tgapi-util-get-last-read-file-suffix)`

Appendix B. Boundary Functions

The following boundary functions are available.

> **Note:**
>
> These functions are not supported in a distributed parallel environment.

Feature	Description	Function
Improve surface mesh	Improve marked faces by splitting the longer edge. • Specify the face zones using a `face-zone-id-list`, `face-zone-name-list`, or `face-zone-name-pattern*`. • Specify the `longer-edge-split-ratio` (length ratio for the split created by projecting the opposite node onto the longer edge). The valid range is `0` to `0.5` and the recommended value is `0.25`.	`(tgapi-boundary-split-marked-faces '(face-zone-id-list) longer-edge-split-ratio)`
		`(tgapi-boundary-split-marked-faces '(face-zone-name-list) longer-edge-split-ratio)`
		`(tgapi-boundary-split-marked-faces "face-zone-name-pattern*" longer-edge-split-ratio)`
	Improve marked faces by collapsing face edges. • Specify the face zones using a `face-zone-id-list`, `face-zone-name-list`, or `face-zone-name-pattern*`. • Choose whether to preserve the boundary by setting `preserve-boundary?` to #t or #f. – If `preserve-boundary?` is enabled (#t), specify the `boundary-corner-angle` to be used for marking fixed nodes on the boundary. The recommended value is `20` degrees.	`(tgapi-boundary-collapse-marked-faces '(face-zone-id-list) preserve-boundary? boundary-corner-angle max-collapsible-edge-ratio)`
		`(tgapi-boundary-collapse-marked-faces '(face-zone-name-list) preserve-boundary? boundary-corner-angle max-collapsible-edge-ratio)`
		`(tgapi-boundary-collapse-marked-faces "face-zone-name-pattern*" preserve-boundary? boundary-corner-angle max-collapsible-edge-ratio)`

Feature	Description	Function
	• Set the `max-collapsible-edge-ratio` depending on the improve operation: – To improve general quality, specify a value between `0` to `0.5`. A value of `0.5` is recommended. – To collapse sliver faces, specify a value of `0.1`. **Note:** A value of `0` will result in all marked faces being collapsed. **Tip:** Use this operation when the mesh skewness is greater than 0.7. Also, it is recommended to mark faces and collapse faces by edge ratio incrementally. For example, first mark faces with skewness greater than 0.9 and then collapse marked faces using an edge ratio of 0.1. When the mesh quality is improved, next mark faces with skewness greater than 0.8 and then collapse marked faces using an edge ratio of 0.2; and so on.	
	Improve sliver faces by splitting and collapsing edges.	`(tgapi-boundary-split-and-collapse-sliver-faces '(face-`

Feature	Description	Function
	• Specify the face zones using a `face-zone-id-list`, `face-zone-name-list`, or `face-zone-name-pattern*`. • Specify the height threshold for sliver face improvement. All faces with `sliver-height` less than the specified value will be marked for improvement. Sliver height is the shortest height of the triangular face - the distance between the location of the opposite node and its projection point on the longer edge. • Specify either `"Skewness"` or `"Aspect Ratio"` as the `"sliver-quality-measure"`, and the `sliver-quality` threshold used for marking. • Specify the `longer-edge-split-ratio` (length ratio for the split created by projecting the opposite node onto the longer edge). The valid range is `0` to `0.5` and the recommended value is `0.25`. • Choose whether to preserve the boundary by setting `preserve-boundary?` to `#t` or `#f`. – If `preserve-boundary?` is enabled (`#t`), specify the `boundary-corner-angle` to be used for marking fixed nodes on the boundary. The recommended value is `20` degrees.	`zone-id-list) sliver-height "sliver-quality-measure" sliver-quality longer-edge-split-ratio preserve-boundary? boundary-corner-angle)`
		`(tgapi-boundary-split-and-collapse-sliver-faces '(face-zone-name-list) sliver-height "sliver-quality-measure" sliver-quality longer-edge-split-ratio preserve-boundary? boundary-corner-angle)`
		`(tgapi-boundary-split-and-collapse-sliver-faces "face-zone-name-pattern*" sliver-height "sliver-quality-measure" sliver-quality longer-edge-split-ratio preserve-boundary? boundary-corner-angle)`
Patch	Create a planar surface to patch holes associated with free faces for the face zones specified. The free faces should be continuous for the planar patch to be created. • Specify the face zones with free faces using a `face-zone-id-`	`(tgapi-boundary-fill-planar-holes-using-free-faces '(face-zone-id-list) plane tolerance absolute-tolerance?)`
		`(tgapi-boundary-fill-planar-holes-using-free-faces '(face-zone-name-list) plane`

Feature	Description	Function
	list, face-zone-name-list, or face-zone-name-pattern*. • Specify three coplanar points to define the plane (for example, (list '(-10 5 5) '(0 5 5) '(0 -5 5))). • Specify the tolerance value, and choose whether to use an absolute tolerance (set absolute-tolerance? to #t or #f). Nodes within the tolerance from the plane will be used to create the planar surface. > **Note:** > > The patch surface is not connected to the original face zones.	tolerance absolute-tolerance?) (tgapi-boundary-fill-planar-holes-using-free-faces "face-zone-name-pattern*" plane tolerance absolute-tolerance?)
	Create a planar surface to patch holes associated with free faces for the object and face zone labels specified. The free faces should be continuous for the planar patch to be created. • Specify the object and the face zone labels with free faces using a face-zone-label-list or face-zone-label-name-pattern*. • Specify three coplanar points to define the plane (for example, (list '(-10 5 5) '(0 5 5) '(0 -5 5))). • Specify the tolerance value, and choose whether to use an absolute tolerance (set absolute-tolerance? to #t or #f). Nodes within the tolerance from the plane will be used to create the planar surface.	(tgapi-boundary-fill-planar-holes-using-free-faces-in-object 'object '(face-zone-label-list) plane tolerance absolute-tolerance? "new-label-name") (tgapi-boundary-fill-planar-holes-using-free-faces-in-object 'object "face-zone-label-name-pattern*" plane tolerance absolute-tolerance? "new-label-name") (tgapi-boundary-fill-planar-holes-using-free-faces-in-object "object" '(face-zone-label-list) plane tolerance absolute-tolerance? "new-label-name") (tgapi-boundary-fill-planar-holes-using-free-faces-in-object "object" "face-zone-label-name-pattern*" plane tolerance absolute-tolerance? "new-label-name")

Feature	Description	Function
	• Specify the `new-label-name` for the planar surface created. The patch surface is included in the object and a new face zone label is created.	
	Create a planar surface to patch holes using the free face edges for the edge zones specified. The edge zone should be continuous and free from intersections. • Specify the edge zones using an `edge-zone-id-list`, `edge-zone-name-list`, or `edge-zone-name-pattern*`.	`(tgapi-boundary-create-planar-surface-using-edges '(edge-zone-id-list) plane tolerance absolute-tolerance?)`
		`(tgapi-boundary-create-planar-surface-using-edges '(edge-zone-name-list) plane tolerance absolute-tolerance?)`
	• Specify three coplanar points to define the `plane` (for example, `(list '(-10 5 5) '(0 5 5) '(0 -5 5)))`. • Specify the `tolerance` value, and choose whether to use an absolute tolerance (set `absolute-tolerance?` to `#t` or `#f`). Nodes within the tolerance from the plane will be used to create the planar surface. **Note:** The patch surface is not connected to the original face zones.	`(tgapi-boundary-create-planar-surface-using-edges "edge-zone-name-pattern*" plane tolerance absolute-tolerance?)`
Remesh	Remesh the marked faces. • Specify the face zones using a `face-zone-id-list`, `face-zone-name-list`, or `face-zone-name-pattern*`. • Specify the number of rings (`n-rings`). • Choose whether to preserve the shared zone boundary by setting `preserve-shared-zone-`	`(tgapi-boundary-remesh-marked-faces '(face-zone-id-list) n-rings preserve-shared-zone-boundary? feature-min-angle feature-max-angle corner-angle "sizing-option" constant-size min-size max-size growth-rate)`
		`(tgapi-boundary-remesh-marked-faces '(face-zone-name-list) n-rings preserve-shared-zone-boundary? fea-`

Feature	Description	Function

	boundary? to #t or #f (recommended).
	
	• Specify the `feature-min-angle` and `feature-max-angle`. The `feature-min-angle` and `feature-max-angle` are the limits for the feature angle. Recommended values are 40 and 180, respectively. If the mesh has false features like folds and overlaps, the `feature-max-angle` can be set to 155.
	
	• Specify the `corner-angle`. The `corner-angle` is the minimum angle between feature edges that will be preserved during remeshing. The recommended value is 20.
	
	• Select a `"sizing-option"` using `"geometric"`, `"constant"`, `"geodesic"` or `"volumetric"`.
	
	• Specify values for `constant-size` (used when `"sizing-option"` is set to `"constant"`), `min-size`, `max-size`, and `growth-rate` (these are used when `"sizing-option"` is set to `"geometric"` or `"geodesic"`).
	
	Note:
	
	All four values must be provided regardless of the selected `"sizing-option"`.
	
	A value of -1 can be used for the parameters that are not required
		`ture-min-angle feature-max-angle corner-angle "sizing-option" constant-size min-size max-size growth-rate)`
		
		`(tgapi-boundary-remesh-marked-faces "face-zone-name-pattern*" n-rings preserve-shared-zone-boundary? fea-ture-min-angle feature-max-angle corner-angle "sizing-option" constant-size min-size max-size growth-rate)`

Feature	Description	Function
	for a particular sizing option.	
	Remesh face zones by separating them internally into patches and later merging them. The patches which fail to remesh using the advancing front (surfer) algorithm will be remeshed using explicit remeshing (split, improve, and coarsen). You can specify the number of attempts for the advancing front algorithm and also use explicit remeshing for the patches, if required.	`(tgapi-boundary-remesh-face-zones '(face-zone-list) '(edge-zone-list) replace-face-zones? feature-min-angle feature-max-angle corner-angle "sizing-option" constant-size min-size max-size growth-rate n-retries-on-failed improve-failed? front-intersection-tolerance)`
	• Specify the face zones and edge zones using a face/edge zone list or by face/edge zone name pattern.	`(tgapi-boundary-remesh-face-zones '(face-zone-list) "edge-zone-name-pattern*" replace-face-zones? feature-min-angle feature-max-angle corner-angle "sizing-option" constant-size min-size max-size growth-rate n-retries-on-failed improve-failed? front-intersection-tolerance)`
	• Choose whether to replace the original zones with the remeshed zones by setting `replace-face-zones?` to `#t` or `#f` (recommended).	
	• Specify the `feature-min-angle` and `feature-max-angle`. The `feature-min-angle` and `feature-max-angle` limits are used to retain angle-based features. Recommended values are `40` and `180`, respectively. If the mesh has false features like folds and overlaps, the `feature-max-angle` can be set to `155`.	`(tgapi-boundary-remesh-face-zones "face-zone-name-pattern*" '(edge-zone-list) replace-face-zones? feature-min-angle feature-max-angle corner-angle "sizing-option" constant-size min-size max-size growth-rate n-retries-on-failed improve-failed? front-intersection-tolerance)`
	• Specify the `corner-angle`. The `corner-angle` is the minimum angle between feature edges that will be preserved for corner nodes during remeshing. The recommended value is `20`.	`(tgapi-boundary-remesh-face-zones "face-zone-name-pattern*" "edge-zone-name-pattern*" replace-face-zones? feature-min-angle feature-max-angle corner-angle "sizing-option" constant-size min-size max-size growth-rate n-retries-on-failed improve-`

Feature	Description	Function
	• Select a "sizing-option" using "constant", "geodesic" or "volumetric". **Note:** The "geometric" option is not supported. • Specify values for constant-size (used when "sizing-option" is set to "constant"), min-size, max-size, and growth-rate (these are used when "sizing-option" is set to "geodesic"). These values are not used when the "sizing-option" is set to "volumetric". **Note:** All four values must be provided regardless of the selected "sizing-option". A value of -1 can be used for the parameters that are not required for a particular sizing option. • Specify the number of attempts to improve the failed patches using explicit remeshing (n-retries-on-failed) and whether to further improve the failed patches using explicit remeshing improve-failed? set to #t or #f. • Specify the front-intersection-tolerance for checking	failed? front-intersection-tolerance)

Feature	Description	Function
	intersections. The value specified should be in the range 0 to 1. The recommended value is `0.0`, which can be increased if the remeshed patch shows intersections.	
	Remesh object face zone labels by separating them internally into patches and later merging them. The patches which fail to remesh using the surfer (advancing front) algorithm, will be remeshed using explicit remeshing (split, improve, and coarsen). You can specify the number of attempts for the surfer algorithm and also use explicit remeshing for the patches, if required.	`(tgapi-boundary-remesh-ob-ject-labels 'object-name '(face-zone-label-list) '(edge-zone-list) feature-min-angle feature-max-angle corner-angle "sizing-option" constant-size min-size max-size growth-rate n-retries-on-failed improve-failed? front-intersection-tolerance)`
	• Specify the object, face zone labels (`face-zone-label-list` or `face-zone-label-pattern*`), and edge zones (`edge-zone-list` or `edge-zone-name-pattern*`). • Specify the `feature-min-angle` and `feature-max-angle`. The `feature-min-angle` and `feature-max-angle` limits are used to retain angle-based features. Recommended values are `40` and `180`, respectively. If the mesh has false features like folds and overlaps, the `feature-max-angle` can be set to `155`.	`(tgapi-boundary-remesh-ob-ject-labels 'object-name '(face-zone-label-list) "edge-zone-name-pattern*" feature-min-angle feature-max-angle corner-angle "siz-ing-option" constant-size min-size max-size growth-rate n-retries-on-failed improve-failed? front-intersection-tolerance)`
		`(tgapi-boundary-remesh-ob-ject-labels 'object-name "face-zone-label-pattern*" '(edge-zone-list) feature-min-angle feature-max-angle corner-angle "sizing-option" constant-size min-size max-size growth-rate n-retries-on-failed improve-failed? front-intersection-tolerance)`
	• Specify the `corner-angle`. The `corner-angle` is the minimum angle between feature edges that will be preserved for corner nodes during remeshing. The recommended value is `20`.	`(tgapi-boundary-remesh-ob-ject-labels "object-name" '(face-zone-label-list) '(edge-zone-list) feature-`

Feature	Description	Function

	• Select a `"sizing-option"` using `"constant"`, `"geodesic"` or `"volumetric"`.
	
	Note:
	
	The `"geometric"` option is not supported.
	
	• Specify values for `constant-size` (used when `"sizing-option"` is set to `"constant"`), `min-size`, `max-size`, and `growth-rate` (these are used when `"sizing-option"` is set to `"geodesic"`). These values are not used when the `"sizing-option"` is set to `"volumetric"`.
	
	Note:
	
	All four values must be provided regardless of the selected `"sizing-option"`.
	
	A value of `-1` can be used for the parameters that are not required for a particular sizing option.
	
	• Specify the number of attempts to improve the failed patches using explicit remeshing (`n-retries-on-failed`) and whether to further improve the failed patches using explicit remeshing `improve-failed?` set to `#t` or `#f`.
	
	• Specify the `front-intersection-tolerance` for checking
		`min-angle feature-max-angle corner-angle "sizing-option" constant-size min-size max-size growth-rate n-retries-on-failed improve-failed? front-intersection-tolerance)`
		
		`(tgapi-boundary-remesh-object-labels "object-name" '(face-zone-label-list) "edge-zone-name-pattern*" feature-min-angle feature-max-angle corner-angle "sizing-option" constant-size min-size max-size growth-rate n-retries-on-failed improve-failed? front-intersection-tolerance)`
		
		`(tgapi-boundary-remesh-object-labels "object-name" "face-zone-label-pattern*" '(edge-zone-list) feature-min-angle feature-max-angle corner-angle "sizing-option" constant-size min-size max-size growth-rate n-retries-on-failed improve-failed? front-intersection-tolerance)`

Feature	Description	Function
	intersections. The value specified should be in the range 0 to 1. The recommended value is `0.0`, which can be increased if the remeshed patch shows intersections.	

B.1. Examples

To improve mesh quality on all face zones, use marking utilities with functions for improving the surface mesh and remeshing.

- Mark faces based on quality:

```
(tgapi-util-mark-faces-by-quality "*" "Skewness" 0.6 #f)
```

- Improve the marked faces by splitting the longer edge.

```
(tgapi-boundary-split-marked-faces "*" 0.25)
```

- Mark faces based on quality:

```
(tgapi-util-mark-faces-by-quality "*" "Skewness" 0.95 #f)
```

- Improve sliver faces

```
(tgapi-boundary-split-and-collapse-sliver-faces "*" 0.01 "Skewness" 0.95
0.25 #t 20)
```

- Mark face strips between all boundaries based on strip height and quality.

```
(tgapi-util-mark-face-strips-by-height-and-quality "*" 3 0.01 "Skewness"
0.9 40)
```

- Improve the marked faces by collapsing face edges.

```
(tgapi-boundary-collapse-marked-faces "*" #f 20 0.5)
```

- Mark faces based on quality:

```
(tgapi-util-mark-faces-by-quality "*" "Skewness" 0.6 #f)
```

- Remesh the marked faces based on the size field:

```
(tgapi-boundary-remesh-marked-faces "*" 3 #f 40 180 20 "geodesic" 0.05
0.05 2.5 1.6)
```

For a mesh object **_fluid**, remesh all face zones using different sizing options:

- Using geodesic sizing

```
(tgapi-boundary-remesh-face-zones (tgapi-util-get-face-zone-id-list-of-
object "_fluid") (tgapi-util-get-edge-zone-id-list-of-object "_fluid")
#t 40 155 20 "geodesic" -1 0.35 260 1.35 1 #t 0.3)
```

- Using volumetric sizing

```
(tgapi-boundary-remesh-face-zones (tgapi-util-get-face-zone-id-list-of-
object "_fluid") (tgapi-util-get-edge-zone-id-list-of-object "_fluid")
#t 40 155 20 "volumetric" -1 -1 -1 -1 1 #t 0.0)
```

- Using constant sizing

```
(tgapi-boundary-remesh-face-zones (tgapi-util-get-face-zone-id-list-of-
object "_fluid") (tgapi-util-get-edge-zone-id-list-of-object "_fluid")
#t 40 155 20 "constant" 2.0 -1 -1 -1 1 #t 0.0)
```

> **Note:**
>
> In these examples, a value of -1 has been used for the parameters that are not required for a particular sizing option.

For a mesh object **_fluid**, remesh object face zone labels using different sizing options:

- Using geodesic sizing

```
(tgapi-boundary-remesh-object-labels "_fluid" "*" "*" 40 155 20
"geodesic" -1 0.5 5 1.2 0 #f 0.0)
```

- Using volumetric sizing

```
(tgapi-boundary-remesh-object-labels "_fluid" "*" "*" 40 155 20 "volu-
metric" -1 -1 -1 -1 1 #f 0.0)
```

- Using constant sizing

```
(tgapi-boundary-remesh-object-labels "_fluid" "*" "*" 40 155 20 "con-
stant" 2.0 -1 -1 -1 1 #f 0.0)
```

> **Note:**
>
> In these examples, a value of -1 has been used for the parameters that are not required for a particular sizing option.

Patch holes associated with free faces.

- Create a planar surface to patch holes associated with free faces for the list of face zones.

```
(tgapi-boundary-fill-planar-holes-using-free-faces '(6 5 3 4) (list '(-
10 5 5) '(0 5 5) '(0 -5 5)) 0.001 #t)
```

- Create a planar surface to patch holes associated with free faces for the list of object face zone labels.

```
(tgapi-boundary-fill-planar-holes-using-free-faces-in-object "object"
"*" (list '(-10 5 5) '(0 5 5) '(0 -5 5)) 0.001 "symmetry" #t)
```

- Create a planar surface to patch holes using the free face edges for the edge zones specified.

```
(tgapi-boundary-create-planar-surface-using-edges "*" (list '(-10 5 5)
'(0 5 5) '(0 -5 5)) 0.001 #f)
```

Appendix C. Connect Functions

The following connect functions are available.

> **Note:**
>
> These functions are not supported in a distributed parallel environment.

Feature Description	Function

Connect overlapping face zone label pairs by joining zones across label pairs.
 	
• Specify the mesh `object-name`.
 	
• Specify a list of face zone `label-pairs` to be connected. For example, `(label1 label2)`.
 	
• Specify values for the parameters:
 	
  – `feature-angle` (default is `40`)
 	
  – `tolerance` (default is `0.95`)
 	
  – `absolute-tolerance?` Specify `#t` or `#f` (default, for relative tolerance).
 	
  – `separate-overlaps?` Specify `#t` (default) or `#f` to separate overlapping zones.
 	
  – `"separate-method"` Specify `"keep-none"` or `"keep-one"` (recommended).
 	
  – `remesh?` Specify `#t` (default) or `#f` to remesh after the connect operation.
 	
  – `"sizing-method"` Set to `"geometric"` (default) or `"size-field"`.
 	
  – `no-of-layers` Specify the number of rings of faces to be remeshed around the connections, default is `3`.
	`(tgapi-connect-join-across-label-pairs 'object-name '(label-pairs) feature-angle tolerance absolute-tolerance? separate-overlaps? "separate-method" remesh? "sizing-method" no-of-layers)`

Connect overlapping face zone label pairs by intersecting zones across label pairs.	`(tgapi-connect-intersect-across-label-pairs 'object-name '(label-pairs)`

Feature Description	Function
• Specify the mesh `object-name`. • Specify a list of face zone `label-pairs` to be connected. For example, (`label1 label2`). • Specify values for the parameters: – `feature-angle` (default is `40`) – `tolerance` (default is `0.95`) – `absolute-tolerance?` Specify `#t` or `#f` (default, for relative tolerance). – `separate-intersections?` Specify `#t` (default) or `#f` to separate intersecting zones. – `remesh?` Specify `#t` (default) or `#f` to remesh after the connect operation. – `"sizing-method"` Set to `"geometric"` (default) or `"size-field"`. – `no-of-layers` Specify the number of rings of faces to be remeshed around the connections, default is `3`.	`feature-angle tolerance absolute-tolerance? separate-intersections? remesh? "sizing-method" no-of-layers)`
Connect overlapping face zone label pairs by stitching zones across label pairs. • Specify the mesh `object-name`. • Specify a list of face zone `label-pairs` to be connected. For example, (`label1 label2`). • Specify values for the parameters: – `"stitch-method"` Specify `"all-all"` (default), `"free-free"`, or `"free-non-free"`. – `feature-angle` (default is `40`) – `tolerance` (default is `0.95`) – `absolute-tolerance?` Specify `#t` or `#f` (default, for relative tolerance).	`(tgapi-connect-stitch-across-label-pairs 'object-name '(label-pairs) "stitch-method" feature-angle tolerance absolute-tolerance? remesh? "sizing-method" no-of-layers)`

234

Feature Description	Function
– `remesh?` Specify `#t` (default) or `#f` to remesh after the connect operation. – `"sizing-method"` Set to `"geometric"` (default) or `"size-field"`. – `no-of-layers` Specify the number of rings of faces to be remeshed around the connections, default is 3.	

C.1. Examples

For a mesh object **_fluid** with face zone labels **part1**, **part2**, and **part3**:

- Connect overlapping face zone label pairs by joining zones across label pairs.

```
(tgapi-connect-join-across-label-pairs '_fluid (list '(part1 part2)
'(part1 part3) '(part2 part3)) 40 0.95 #f #t "keep-one" #t "geometric"
3)
```

- Connect overlapping face zone label pairs by intersecting zones across label pairs.

```
(tgapi-connect-intersect-across-label-pairs '_fluid (list '(part1 part2)
'(part1 part3) '(part2 part3)) 40 0.95 #f #t #t "geometric" 3)
```

- Connect overlapping face zone label pairs by stitching zones across label pairs.

```
(tgapi-connect-stitch-across-label-pairs '_fluid (list '(part1 part2)
'(part1 part3) '(part2 part3)) "all-all" 40 0.95 #f #t "geometric" 3)
```

Appendix D. Size Field Functions

The size field defines the sizes in the volume. Having volumetric sizes could pose problems where sizes are diffused across dead space, leading to unnecessary refinement. In addition to volumetric sizing, a new type of size definition (geodesic) is introduced to define the sizes along a surface rather than the volume. The geodesic sizing enables you to confine sizes to surfaces and avoid problems like dead space refinement.

The following size field functions are available.

> **Note:**
>
> These functions are not supported in a distributed parallel environment.

Feature		Description	Function
Initialize geodesic sizing	On face zones	Initialize geodesic sizing on the specified face zones. • Specify the face zones using a `face-zone-id-list`, `face-zone-name-list`, or `face-zone-name-pattern*`. • Specify the `max-size` value.	`(tgapi-size-field-initialize-geodesic-on-face-zones '(face-zone-name-list) max-size)`
			`(tgapi-size-field-initialize-geodesic-on-face-zones '(face-zone-ID-list) max-size)`
			`(tgapi-size-field-initialize-geodesic-on-face-zones "face-zone-name-pattern*" max-size)`
	On object face zone labels	Initialize geodesic sizing on the specified object face zone labels. • Specify the `object-list` or `object-name-pattern*` and the face zone labels using a `face-zone-label-list` or `face-zone-label-name-pattern*`. • Specify the `max-size` value.	`(tgapi-size-field-initialize-geodesic-on-object-labels '(object-list) '(face-zone-label-list) max-size)`
			`(tgapi-size-field-initialize-geodesic-on-object-labels '(object-list) "face-zone-label-name-pattern*" max-size)`
			`(tgapi-size-field-initialize-geodesic-on-object-labels "object-name-pattern*" '(face-zone-label-list) max-size)`

Feature		Description	Function
			`(tgapi-size-field-initialize-geodesic-on-object-labels "object-name-pattern*" "face-zone-label-name-pattern*" max-size)`
	On object face zones	Initialize geodesic sizing on the face zones for the specified objects. • Specify the `object-list` or `object-name-pattern*`. • Specify the `max-size` value.	`(tgapi-size-field-initialize-geodesic-on-objects '(object-list) max-size)`
			`(tgapi-size-field-initialize-geodesic-on-objects "object-name-pattern*" max-size)`
Compute size field	Geodesic sizes	Computes geodesic sizes on face nodes based on the size controls specified.	`(tgapi-size-field-compute-geodesic '(size-control-name-list))`
			`(tgapi-size-field-compute-geodesic '(size-control-name-pattern*))`
			`(tgapi-size-field-compute-geodesic "size-control-name-pattern*)`
	Volumetric sizes	Computes the volumetric size field based on the size controls specified.	`(tgapi-size-field-compute-volumetric '(size-control-name-list))`
			`(tgapi-size-field-compute-volumetric '(size-control-name-pattern*))`
			`(tgapi-size-field-compute-volumetric "size-control-name-pattern*")`
	Geodesic blended size field	Computes the geodesic blended size field as follows. • The volumetric size field is computed based on all edge, face proximity and BOI controls. • Geodesic sizes are computed based on the remaining size controls. • Updates geodesic sizes based on volumetric sizes (the lower minimum size (geodesic or volumetric) will be used). • Hard sizes are computed last.	`(tgapi-size-field-compute-geodesic-blended '(size-control-name-list))`
			`(tgapi-size-field-compute-geodesic-blended '(size-control-name-pattern*))`
			`(tgapi-size-field-compute-geodesic-blended "size-control-name-pattern*")`

Feature		Description	Function
Update geodesic sizes with volumetric sizes	On face zones	Updates nodal (geodesic) sizes based on the existing volumetric size field for the face zones specified. This requires the volumetric size field to be computed. Specify the face zones using a `face-zone-id-list`, `face-zone-name-list`, or `face-zone-name-pattern*`. **Note:** Geodesic sizes need to be initialized and computed before attempting an update.	`(tgapi-size-field-update-geodesic-with-volumetric-on-face-zones '(face-zone-id-list))`
			`(tgapi-size-field-update-geodesic-with-volumetric-on-face-zones '(face-zone-name-list))`
			`(tgapi-size-field-update-geodesic-with-volumetric-on-face-zones "face-zone-name-pattern*")`
	On object face zone labels	Updates nodal (geodesic) sizes based on the existing volumetric size field for the face zone labels specified. This requires the volumetric size field to be computed. Specify the `object-name` and the face zone labels using a `face-zone-label-list` or `face-zone-label-name-pattern*`. **Note:** Geodesic sizes need to be initialized and computed before attempting an update.	`(tgapi-size-field-update-geodesic-with-volumetric-on-object-labels 'object-name '(face-zone-label-list))`
			`(tgapi-size-field-update-geodesic-with-volumetric-on-object-labels 'object-name '(face-zone-label-name-pattern*))`
			`(tgapi-size-field-update-geodesic-with-volumetric-on-object-labels 'object-name "face-zone-label-name-pattern*")`
			`(tgapi-size-field-update-geodesic-with-volumetric-on-object-labels "object-name" '(face-zone-label-list))`
			`(tgapi-size-field-update-geodesic-with-volumetric-on-object-labels "object-name" '(face-zone-label-name-pattern*))`
			`(tgapi-size-field-update-geodesic-with-volumetric-on-object-labels "object-name" "face-zone-label-name-pattern*")`

239

Feature		Description	Function
	On object face zones	Updates nodal (geodesic) sizes based on the existing volumetric size field for the face zones of the objects specified. This requires the volumetric size field to be computed. Specify the `object-list` or `object-name-pattern*`. **Note:** Geodesic sizes need to be initialized and computed before attempting an update.	`(tgapi-size-field-update-geodesic-with-volumetric-on-objects '(object-list))`
			`(tgapi-size-field-update-geodesic-with-volumetric-on-objects "object-name-pattern*")`
Scale geodesic sizes	On marked faces	Scales and limits geodesic sizes on the marked faces for the face zones specified. • Specify the face zones using a `face-zone-id-list`, `face-zone-name-list`, or `face-zone-name-pattern*`. • Specify the `scale-factor`. • Specify the values for `min-size` and `max-size`	`(tgapi-size-field-scale-geodesic-on-marked-faces '(face-zone-id-list) scale-factor min-size max-size)`
			`(tgapi-size-field-scale-geodesic-on-marked-faces '(face-zone-name-list) scale-factor min-size max-size)`
			`(tgapi-size-field-scale-geodesic-on-marked-faces "face-zone-name-pattern*" scale-factor min-size max-size)`
	On face zones	Scales and limits geodesic sizes on the faces for the face zones specified. • Specify the face zones using a `face-zone-id-list`, `face-zone-name-list`, or `face-zone-name-pattern*`. • Specify the `scale-factor`. • Specify the values for `min-size` and `max-size`	`(tgapi-size-field-scale-geodesic-on-face-zones '(face-zone-id-list) scale-factor min-size max-size)`
			`(tgapi-size-field-scale-geodesic-on-face-zones '(face-zone-name-list) scale-factor min-size max-size)`
			`(tgapi-size-field-scale-geodesic-on-face-zones "face-zone-name-pattern*" scale-factor min-size max-size)`

Feature		Description	Function
	On object face zone labels	Scales and limits geodesic sizes on the faces for the face zone labels specified. • Specify the `object-name` and the face zone labels using a `face-zone-label-list` or `face-zone-label-name-pattern*`. • Specify the `scale-factor`. • Specify the values for `min-size` and `max-size`	`(tgapi-size-field-scale-geodesic-on-object-labels 'object-name '(face-zone-la-bel-list) scale-factor min-size max-size)`
			`(tgapi-size-field-scale-geodesic-on-object-labels 'object-name "face-zone-la-bel-name-pattern*" scale-factor min-size max-size)`
			`(tgapi-size-field-scale-geodesic-on-object-labels "object-name" '(face-zone-label-list) scale-factor min-size max-size)`
			`(tgapi-size-field-scale-geodesic-on-object-labels "object-name" "face-zone-la-bel-name-pattern*" scale-factor min-size max-size)`
	On object face zones	Scales and limits geodesic sizes on the object face zones specified. • Specify the `object-list` or `object-name-pattern*`. • Specify the `scale-factor`. • Specify the values for `min-size` and `max-size`	`(tgapi-size-field-scale-geodesic-on-objects '(ob-ject-list) '(face-zone-la-bel-list) scale-factor min-size max-size)`
			`(tgapi-size-field-scale-geodesic-on-object-labels "object-name-pattern*" "face-zone-label-name-pat-tern*" scale-factor min-size max-size)`
Filter noise	On face zones	Removes refined surface mesh clusters generated by geodesic sizing on the specified face zones. • Specify the face zones using a `face-zone-id-list`, `face-zone-name-list`, or `face-zone-name-pattern*`. • Specify the `growth-rate`. It is recommended to use a growth rate value higher than the value used for computing the geodesic sizes (for example, `1.6`). • Specify the `face-count` threshold. For face clusters	`(tgapi-size-field-filter-noise-geodesic '(face-zone-id-list) growth-rate face-count)`
			`(tgapi-size-field-filter-noise-geodesic '(face-zone-name-list) growth-rate face-count)`
			`(tgapi-size-field-filter-noise-geodesic "face-zone-name-pattern*" growth-rate face-count)`

241

Feature		Description	Function
		whose count is less than the specified threshold value, the geodesic size values will be set to the average of the values on the cluster boundary. **Note:** Geodesic sizes need to be initialized and computed, but undiffused for the face zones specified.	

Diffuse geodesic sizing	On face zones	Diffuse geodesic sizing on the specified face zones. Diffusion can also happen on face zones which are not specified, if they can be reached by node flood filling.
		
		• Specify the face zones using a `face-zone-id-list`, `face-zone-name-list`, or `face-zone-name-pattern*`.
		
		• Specify the minimum and maximum size limits to apply for the node size (`min-size`, `max-size`.
		
		Important:
		
		The sizes will be bounded to the `min-size` and `max-size` specified. The final minimum node size will be the maximum of (node-min-size, min-size) and final maximum node size will be the minimum of (node-max-size, max-size).
		
		It is recommended that you specify the global minimum size
			`(tgapi-size-field-diffuse-geodesic-on-face-zones '(face-zone-name-list) min-size max-size growth-rate diffuse-across-free-edge? free-diffusion-range)`
			
			`(tgapi-size-field-diffuse-geodesic-on-face-zones '(face-zone-id-list) min-size max-size growth-rate diffuse-across-free-edge? free-diffusion-range)`
			
			`(tgapi-size-field-diffuse-geodesic-on-face-zones "face-zone-name-pattern*" min-size max-size growth-rate diffuse-across-free-edge? free-diffusion-range)`

Feature		Description	Function
		required, as you cannot revert to the earlier minimum size after size diffusion.	
		• Specify the `growth-rate` to be applied to diffuse size geodesically (along the surface).	
		• Choose whether to enable diffusion of free node sizes to other free nodes in proximity, even when they are not connected (set `diffuse-across-free-edge?` to `#t` or `#f`).	
		• Specify the absolute range to be used to find free nodes in proximity for diffusion `free-diffusion-range` (when `diffuse-across-free-edge?` is enabled (`#t`).	
	On object face zone labels	Diffuse geodesic sizing on the specified object labels. Diffusion can also happen on face zone labels which are not specified, if they can be reached by node flood filling. • Specify the object face zone labels using the `object-name` and the `label-name-list` or `label-name-pattern*`. • Specify the minimum and maximum size limits to apply for the node size (`min-size`, `max-size`.	`(tgapi-size-field-diffuse-geodesic-on-object-labels 'object-name '(label-name-list) min-size max-size growth-rate diffuse-across-free-edge? free-diffusion-range)`
			`(tgapi-size-field-diffuse-geodesic-on-object-labels 'object-name "label-name-pattern*" min-size max-size growth-rate diffuse-across-free-edge? free-diffusion-range)`
		Important: The sizes will be bounded to the `min-size` and `max-size` specified. The final minimum node size	`(tgapi-size-field-diffuse-geodesic-on-object-labels "object-name" '(label-name-list) min-size max-size growth-rate diffuse-across-free-edge? free-diffusion-range)`
			`(tgapi-size-field-diffuse-geodesic-on-object-labels`

Feature	Description	Function
	will be the maximum of (node-min-size, min-size) and final maximum node size will be the minimum of (node-max-size, max-size). • Specify the `growth-rate` to be applied to diffuse size geodesically (along the surface). • Choose whether to enable diffusion of free node sizes to other free nodes in proximity, even when they are not connected (set `diffuse-across-free-edge?` to #t or #f). • Specify the absolute range to be used to find free nodes in proximity for diffusion `free-diffusion-range` (when `diffuse-across-free-edge?` is enabled (#t).	`"object-name" "label-name-pattern*" min-size max-size growth-rate diffuse-across-free-edge? free-diffusion-range)`

D.1. Examples

Consider a model with face zones **part1**, **part2**, **part3**, **inlet1**, **inlet2**, and **outlet**.

• Initialize geodesic sizes for the face zones.

```
(tgapi-size-field-initialize-geodesic-on-face-zones '(inlet1 outlet)
32.0)

(tgapi-size-field-initialize-geodesic-on-face-zones "part*" 64.0)
```

• Compute the geodesic sizes for curvature size controls.

```
(tgapi-size-field-compute-geodesic 'curvature*)
```

• Compute the volumetric sizes for BOI size controls.

```
(tgapi-size-field-compute-volumetric 'boi*)
```

Compute the geodesic blended size field.

```
(tgapi-size-field-compute-geodesic-blended '*)
```

• Remove refined surface mesh clusters generated by geodesic sizing on the specified face zones.

```
(tgapi-size-field-filter-noise-geodesic "part*" 1.6 40)
```

• Update geodesic sizes with volumetric sizes on selected object face zone labels.

```
(tgapi-size-field-update-geodesic-with-volumetric-on-face-zones 'part*)
```

• Scale geodesic sizes on marked faces for face zones.

```
(tgapi-size-field-scale-geodesic-on-marked-faces '(inlet1 outlet) 0.5
0.15 256)
```

• Scale geodesic sizes on face zones.

```
(tgapi-size-field-scale-geodesic-on-face-zones "part*" 0.7 0.15 256)
```

Consider a model with objects **part1**, **part2**, and **part3**, **inlet1**, **inlet2**, and **outlet**.

• Initialize geodesic sizes for objects.

```
(tgapi-size-field-initialize-geodesic-on-objects '(inlet1 inlet2 outlet)
32.0)
```

```
(tgapi-size-field-initialize-geodesic-on-objects 'part* 64.0)
```

• Compute the geodesic sizes for curvature size controls.

```
(tgapi-size-field-compute-geodesic 'curvature*)
```

• Compute the volumetric sizes for BOI size controls.

```
(tgapi-size-field-compute-volumetric 'boi*)
```

Compute the geodesic blended size field.

```
(tgapi-size-field-compute-geodesic-blended '*)
```

• Remove refined surface mesh clusters generated by geodesic sizing on the face zones of mesh objects.

```
(tgapi-size-field-filter-noise-geodesic (get-face-zones-of-objects
(tgapi-util-get-object-name-list-of-type 'mesh)) 1.6 40)
```

• Update geodesic sizes with volumetric sizes on selected face zones.

```
(tgapi-size-field-update-geodesic-with-volumetric-on-objects '(inlet1
inlet2 outlet))
```

• Scale geodesic sizes on objects.

```
(tgapi-size-field-scale-geodesic-on-objects 'part* 0.5 0.15 256)
```

Consider a model with mesh object **_fluid** with face zone labels **part1**, **part2**, and **part3**, **inlet1**, **inlet2**, and **outlet**.

- Initialize geodesic sizes for object face zone labels.

  ```
  (tgapi-size-field-initialize-geodesic-on-object-labels '_fluid '(inlet1
  outlet) 32.0)
  ```

  ```
  (tgapi-size-field-initialize-geodesic-on-object-labels '_fluid "part*"
  64.0)
  ```

- Compute the geodesic sizes for curvature size controls.

  ```
  (tgapi-size-field-compute-geodesic 'curvature*)
  ```

- Compute the volumetric sizes for BOI size controls.

  ```
  (tgapi-size-field-compute-volumetric 'boi*)
  ```

  Compute the geodesic blended size field.

  ```
  (tgapi-size-field-compute-geodesic-blended '*)
  ```

- Update geodesic sizes with volumetric sizes on selected object face zone labels.

  ```
  (tgapi-size-field-update-geodesic-with-volumetric-on-object-labels
  '_fluid '(inlet1 outlet))
  ```

- Scale geodesic sizes on object face zone labels.

  ```
  (tgapi-size-field-scale-geodesic-on-object-labels '_fluid "part*" 0.7
  0.15 256)
  ```

Appendix E. Wrap Functions

Wrapping functions are available for generating the surface mesh for automotive applications for underhood thermal management (UTM) and external-aero analysis. Note that periodic meshes are not supported.

The following wrap functions are available.

> **Note:**
>
> These functions are not supported in a distributed parallel environment.

Feature	Description	Function
Wrap objects	Creates the wrapper surface based on the objects and material point specified. The wrapper surface creation involves operations such as Octree creation, intersection, interface creation, projection and zone separation. • Specify the objects using an `object-list` or `object-name-pattern*`. • Specify a name for the new object. • Specify the material point (`"material-point-name"`) to be used for the wrap operation. • Specify the sizing type (set `"sizing-type"` to `"geodesic"` or `"volumetric"`. > **Note:** > > The size field should be computed prior to invoking this API.	`(tgapi-wrap-objects '(object-list) 'new-object-name "material-point-name" "sizing-type")`
		`(tgapi-wrap-objects '(object-list) "new-object-name" "material-point-name" "sizing-type")`
		`(tgapi-wrap-objects "object-name-pattern*" 'new-object-name "material-point-name" "sizing-type")`
		`(tgapi-wrap-objects "object-name-pattern*" "new-object-name" "material-point-name" "sizing-type")`
Wrap objects with seals	Similar to `tgapi-wrap-objects` with the addition of seal objects that are used during interface creation and projection	`(tgapi-wrap-objects-with-seals object-name-list-or-pattern 'new-object-name "material-`

Feature	Description	Function
	only, in order to prevent leaks. Seal objects do not participate in imprinting and are not associated with the underlying geometry. • Specify the objects using an `ob-ject-name` (string or symbol) or `object-list` (string or symbol) or `object-name-pattern*` (string or symbol). • Specify a name for the new object. • Specify the material point (`"ma-terial-point-name"`) to be used for the wrap operation. • Specify the sizing type (set `"siz-ing-type"` to `"geodesic"` or `"volumetric"`. **Note:** The size field should be computed prior to invoking this API. • Specify the seal objects using an `object-name` (string or symbol) or `object-list` (string or symbol) or `object-name-pat-tern*` (string or symbol).	`point-name" "sizing-type" seal-object-name-list-or-pat-tern)`

Imprint object features	Imprint feature edges on given mesh object, and returns feature path extracted after imprint.
	
	• Specify the mesh object using the `mesh-object-name`.
	
	• Choose whether to delete feature path edges far from geometry. Set `separate-off-geometry-edges?` to #t or #f.
	
	• Specify the distance tolerance (`relative-distance`) and angle tolerance (`critical-angle`) to determine the features
		`(tgapi-wrap-imprint-object-features 'mesh-object-name separate-off-geometry-edges? relative-distance critical-angle)`
		
		`(tgapi-wrap-imprint-object-features "mesh-object-name" separate-off-geometry-edges? relative-distance critical-angle)`

Feature		Description	Function
		far from the geometry, when `separate-off-geometry-edges?` is set to `#t`.	
Smooth face nodes	Face zones specified	Smooth face nodes while preserving the features for the face zones specified. • Specify the face zones using a `face-zone-id-list`, `face-zone-name-list`, or `face-zone-name-pattern*`. • Specify the number of mean filter iterations (`mean-filter-iter-ations`). Mean filters use the average of adjacent face normal for smoothing. This method is better for smoothing spikes • Specify the number of median filter iterations (`median-fil-ter-iterations`). Median filters use the median of adjacent face normal for smoothing. This method is better for feature preservation • Specify the number of layers (`number-of-layers`) to be smoothed. **Note:** You can use the mean and median filters individually or use both together.	`(tgapi-wrap-smooth-face-zones-with-features-preserved '(face-zone-id-list) mean-filter-iterations median-fil-ter-iterations number-of-lay-ers)`
			`(tgapi-wrap-smooth-face-zones-with-features-preserved '(face-zone-name-list) mean-filter-iterations median-fil-ter-iterations number-of-lay-ers)`
			`(tgapi-wrap-smooth-face-zones-with-features-preserved "face-zone-name-pattern*" mean-fil-ter-iterations median-filter-iterations number-of-layers)`
	Object face zone labels specified	Smooth face nodes while preserving the features for the object face zone labels specified. • Specify the mesh object and the face zone labels using a `face-zone-label-list` or `face-zone-label-name-pattern*`. • Specify the number of mean filter iterations (`mean-filter-iter-`	`(tgapi-wrap-smooth-object-la-bels-with-features-preserved 'mesh-object-name '(face-zone-label-list) mean-filter-itera-tions median-filter-iterations number-of-layers)`
			`(tgapi-wrap-smooth-object-la-bels-with-features-preserved 'mesh-object-name "face-zone-label-name-pattern*" mean-filter-iterations median-fil-`

Feature		Description	Function
		ations). Mean filters use the average of adjacent face normal for smoothing. This method is better for smoothing spikes • Specify the number of median filter iterations (`median-filter-iterations`). Median filters use the median of adjacent face normal for smoothing. This method is better for feature preservation • Specify the number of layers (`number-of-layers`) to be smoothed. **Note:** You can use the mean and median filters individually or use both together.	`ter-iterations number-of-layers)`
			`(tgapi-wrap-smooth-object-labels-with-features-preserved "mesh-object-name" '(face-zone-label-list) mean-filter-iterations median-filter-iterations number-of-layers)`
			`(tgapi-wrap-smooth-object-labels-with-features-preserved "mesh-object-name" "face-zone-label-name-pattern*" mean-filter-iterations median-filter-iterations number-of-layers)`
	Face nodes far from geometry	Smooth the face nodes of the mesh object which are far from the geometry. Specify the mesh object.	`(tgapi-wrap-smooth-object-off-geometry-faces 'mesh-object-name)`
			`(tgapi-wrap-smooth-object-off-geometry-faces "mesh-object-name")`
Separate object face zones		Separate the mesh object face zones based on the underlying geometry. • Specify the mesh object. • Choose whether to remove saw tooth configurations (set `remove-saw-tooth-configurations?` to `#t` or `#f`). Specify an appropriate value for `saw-tooth-feature-angle`. This is used to identify angle-based features so as to avoid removal of saw-tooth configurations, while separating zones. For example, at the corner of a cuboid formed by six planar zones, a corner face is topologically a saw tooth	`(tgapi-wrap-separate-object-face-zones 'mesh-object-name remove-saw-tooth-configurations? saw-tooth-feature-angle)`
			`(tgapi-wrap-separate-object-face-zones "mesh-object-name" remove-saw-tooth-configurations? saw-tooth-feature-angle)`

Feature	Description	Function
	configuration, but will not be rezoned if the `saw-tooth-fea-ture-angle` specified is less than 90 degree. The recommended range is 30 to 60 degrees.	

Remesh mesh object	Remesh the mesh object based on the sizing type specified.
	
	• Specify the mesh object.
	
	• Specify the `feature-min-angle` and `feature-max-angle`. The `feature-min-angle` and `feature-max-angle` limits are used to retain angle-based features. The recommended value for `fea-ture-max-angle` is 180.
	
	• Specify the `corner-angle`. The `corner-angle` is the minimum angle between feature edges that will be preserved for corner nodes during remeshing.
	
	• Specify the sizing type (set `"siz-ing-type"` to `"geodesic"` or `"volumetric"`).
	
	> **Note:**
	>
	> The size field should be computed prior to invoking this API.
	
	• Specify if you want to respect zone boundaries.
	
	• Specify the `front-intersec-tion-tolerance` for checking intersections. The value specified should be in the range 0 to 1. The recommended value is `0.0`, which can be increased if the remeshed patch shows intersections.
		`(tgapi-wrap-remesh-object 'mesh-object-name feature-min-angle feature-max-angle corner-angle "sizing-type" respect-zone-boundary? front-intersection-tolerance)`
		
		`(tgapi-wrap-remesh-object "mesh-object-name" feature-min-angle feature-max-angle corner-angle "sizing-type" respect-zone-boundary? front-intersection-tolerance)`

Feature		Description	Function
Delete unwetted object faces		Separate and delete the object faces that are not in the region of the given material points. • Specify the mesh object. • Specify the objects using a `material-point-name-list` or `material-point-name-pattern*`.	`(tgapi-wrap-delete-unwetted-object-faces 'mesh-object-name material-point-name-list-or-pattern)`
Resolve face intersections		Resolve face intersections for the mesh object specified.	`(tgapi-wrap-resolve-face-intersections-in-object 'mesh-object-name)`
			`(tgapi-wrap-resolve-face-intersections-in-object "mesh-object-name")`
Inflate object faces	By dihedral angle	Inflate mesh object faces based on the dihedral angle specified. • Specify the mesh object. • Specify an appropriate value for `minimum-dihedral-angle`. The `minimum-dihedral-angle` is the angle between faces connected by an edge. The faces will be inflated if the angle is less than the specified value. • Specify the `absolute-min-offset` and `relative-min-offset`. The `absolute-min-offset` is the absolute distance used to move the nodes to increase the gap. The `relative-min-offset` is the relative distance (as a factor of local edge length) used to move the nodes. • Specify the number of iterations (`number-of-iterations`). • Choose whether to remove saw tooth configurations (set `remove-saw-tooth-configurations?` to #t or #f.	`(tgapi-wrap-inflate-object-faces-by-dihedral-angle 'mesh-object-name minimum-dihedral-angle absolute-min-offset relative-min-offset number-of-iterations remove-saw-tooth-configurations?)`
			`(tgapi-wrap-inflate-object-faces-by-dihedral-angle "mesh-object-name" minimum-dihedral-angle absolute-min-offset relative-min-offset number-of-iterations remove-saw-tooth-configurations?)`

Feature		Description	Function

	By invalid normal	Inflate mesh object faces based on invalid normals.
		
		• Specify the mesh object.
		
		• Specify an appropriate value for `number-of-layers`. The `number-of-layers` is the number of layers of face nodes around the invalid normal location to be inflated.
		
		• Specify the `absolute-min-offset` and `relative-min-offset`.
		
		The `absolute-min-offset` is the absolute distance used to move the nodes to increase the gap. The `relative-min-offset` is the relative distance (as a factor of local edge length) used to move the nodes.
		
		• Specify the number of iterations (`number-of-iterations`).
		
		• Choose whether to remove saw tooth configurations (set `remove-saw-tooth-configurations?` to `#t` or `#f`.
			`(tgapi-wrap-inflate-object-faces-by-invalid-normal 'mesh-object-name number-of-layers absolute-min-offset relative-min-offset number-of-itera-tions remove-saw-tooth-config-urations?)`
			
			`(tgapi-wrap-inflate-object-faces-by-dihedral-angle "mesh-object-name" number-of-layers absolute-min-offset relative-min-offset number-of-itera-tions remove-saw-tooth-config-urations?)`

Improve mesh object	Improve mesh object quality	Improve the mesh object quality.
		
		• Specify the mesh object.
		
		• Enable aggressive improvement if required (set `aggressively?` to `#t` or `#f`. When this option is enabled, the features may not be preserved during the improve operation.
			`(tgapi-wrap-improve-object-quality 'mesh-object-name ag-gressively?)`
			
			`(tgapi-wrap-improve-object-quality "mesh-object-name" aggressively?)`

	For tet meshing	Improve mesh object for tet meshing.
		
		• Specify the mesh object.
		
		• Specify the material point (`"ma-terial-point-name"`).
			`(tgapi-wrap-improve-object-for-tet-meshing 'mesh-object-name "material-point-name" max-cell-skewness max-face-skewness max-face-size-change)`
			
			`(tgapi-wrap-improve-object-for-tet-meshing "mesh-object-`

Feature		Description	Function
		• Set the `max-cell-skewness`, `max-face-skewness`, and `max-face-size-change`.	`name" "material-point-name" max-cell-skewness max-face-skewness max-face-size-change)`
	For prism meshing	Improve mesh object for prism meshing. In general, zones will inherit the prism settings from the underlying geometry. For zones that do not inherit prism settings, the settings passed as arguments will be used. • Specify the mesh object. • Specify the number of layers (`number-of-layers`). The number of layers should be the same as that set for prism meshing. • Set the prism `offset-method`. Select one of the options `"aspect-ratio"`, `"last-ratio"`, `"uniform"`, or `"minimum-height"`. • Set the `prism-aspect-ratio`, if applicable. • Set the `last-ratio-percentage`, if applicable. • Set the `prism-layer-first-height`. • Set the `prism-layer-growth-rate`. • Specify the `quality-measure` to be used. • Set the `max-cell-skewness` and `max-face-skewness`. • Choose whether to retain the prisms (set `keep-prisms?` to #f	`(tgapi-wrap-improve-object-for-prism-meshing 'mesh-object-name number-of-layers "offset-method" prism-aspect-ratio last-ratio-percentage prism-layer-first-height prism-layer-growth-rate quality-measure max-cell-skewness max-face-skewness keep-prisms?)`
			`(tgapi-wrap-improve-object-for-prism-meshing "mesh-object-name" number-of-layers offset-method prism-aspect-ratio last-ratio-percentage prism-layer-first-height prism-layer-growth-rate quality-measure max-cell-skewness max-face-skewness keep-prisms?)`

Feature		Description	Function
		or #t) (setting to #t requires beta features to be enabled). **Note:** A value of -1 can be used for the parameters that are not required for a particular offset method.	

E.1. Examples

For a model with geometry objects **part1**, **part2**, **part3** and material point **fluid**:

- Compute the size field.

```
(tgapi-size-field-compute-geodesic-blended '*)
```

- Wrap the geometry objects using the material point **fluid**.

```
(tgapi-wrap-objects 'part* '_fluid "fluid" "geodesic")
```

- Imprint feature edges on the mesh object.

```
(tgapi-wrap-imprint-object-features '_fluid #t 0.2 15)
```

- Smooth face nodes while preserving features.

```
(tgapi-wrap-smooth-face-zones-with-features-preserved (get-face-zones-
of-objects '(_fluid)) 5 15 0)
```

- Smooth the face nodes of the mesh object which are far from the geometry.

```
(tgapi-wrap-smooth-object-off-geometry-faces '_fluid)
```

- Remesh the mesh object.

```
(tgapi-wrap-remesh-object '_fluid 40.0 180.0 180.0 "geodesic" #f 0.0)
```

- Improve the mesh object quality.

```
(tgapi-wrap-improve-object '_fluid #t)
```

- Resolve thin regions and self intersections.

```
(tgapi-wrap-inflate-object-faces-by-invalid-normal '_fluid 2 0.0 0.2 2
#t)
```

```
(tgapi-wrap-inflate-object-faces-by-dihedral-angle '_fluid 10 0.0 0.2
2 #t)
```

```
(tgapi-wrap-resolve-face-intersections-in-object '_fluid)
```

• Improve mesh object for tet meshing.

```
(tgapi-wrap-improve-object-for-tet-meshing '_fluid "fluid" 0.96 0.8 2.0)
```

• Improve mesh object for prism meshing.

```
(tgapi-wrap-improve-object-for-prism-meshing '_fluid 1 "aspect-ratio"
5 -1 -1 1.2 "inverse-ortho" 0.97 0.9 #f)
```

> **Note:**
>
> A value of -1 has been used for the parameters that are not required for the offset
> method chosen.

• Remove unwetted object faces.

```
(tgapi-wrap-delete-unwetted-object-faces 'wrap-v2 'ext*)
```

```
(tgapi-wrap-delete-unwetted-object-faces "wrap-v2" (list "external"
"mat-pt1"))
```

```
(tgapi-wrap-delete-unwetted-object-faces "wrap-v2" "mat-pt1")
```

• Separate the mesh object face zones based on the geometry.

```
(tgapi-wrap-separate-object-face-zones '_fluid #t 0.0)
```

Part III: Solution Mode

The section describes the text command listing for Fluent in Solution mode.

- `adjoint/` (p. 259) lists commands related to the adjoint solver.

- `define/` (p. 271) lists commands related to problem definition, such as models, boundary conditions, materials, etc.

- `display/` (p. 383) lists commands used to control the rendering of your model and mesh in the graphical window.

- `exit / close-fluent` (p. 411) lists the command to close the program.

- `file/` (p. 413) lists commands used to input and output your data.

- `mesh/` (p. 431) lists commands used to create and manage mesh properties.

- `parallel/` (p. 441) lists commands specific to parallel processing.

- `plot/` (p. 447) lists commands specific to plotting data.

- `preferences/` (p. 453) lists commands for controlling preferences.

- `report/` (p. 465) lists commands used to return statistics for the simulation.

- `server/` (p. 475) lists commands used control the Fluent Remote Visualization Client and Server.

- `solve/` (p. 477) lists commands used to create and manage solution controls, such as animation, cell registers, monitors, initialization, etc.

- `surface/` (p. 517) lists commands related to creating and manipulating surfaces.

- `switch-to-meshing-mode` (p. 521) lists the command to transfer your solution data to Fluent in meshing mode.

- `turbo-post/` (p. 523) lists commands related to results and reporting for turbomachinery.

- `views/` (p. 525) lists commands related to camera and view manipulation in the graphics window.

- Battery Model Text Commands (p. 527) lists commands related to battery models.

- Fuel Cell Text Commands (p. 529) lists commands related to fuel cell models.

- Magnetohydrodynamics Text Commands (p. 541) lists commands related to magnetohydrodynamics models.

- `continuous-fiber/` (p. 543) lists commands related to the continuous fiber model.

Chapter 1: `adjoint/`

`adjoint/`**controls/**

Menu to configure adjoint solver controls.

   `adjoint/controls/`**settings**

   Sets parameters for the adjoint solver numerics.

   `adjoint/controls/`**stabilization**

   Sets parameters for the adjoint stabilization schemes.

   `adjoint/controls/`**stabilization-strategy/**

   Enters the stabilization strategy menu, where you can define the stabilization strategy and settings for the adjoint solver.

      `adjoint/controls/stabilization-strategy/`**complex-case-settings**

      Defines the settings that are recommended for complex cases.

      `adjoint/controls/stabilization-strategy/`**default-settings**

      Defines settings that are recommended for standard cases. These settings are used by default.

      `adjoint/controls/stabilization-strategy/`**settings**

      Allows you to define a custom settings.

      `adjoint/controls/stabilization-strategy/`**type**

      Allows you to define the stabilization strategy type.

`adjoint/`**design-tool/**

Enters the design-tool menu.

   `adjoint/design-tool/`**design-change/**

   Performs optimal shape modification.

      `adjoint/design-tool/design-change/`**calculate-design-change**

      Computes the optimal design change.

`adjoint/design-tool/design-change/`check

Prints a report in the console that summarizes the control points defined for the region, the defined constraining and/or deformation conditions, and any possible conflicts between multiple constraints / deformations applied on a single node.

`adjoint/design-tool/design-change/`display-applied-conditions

Displays all of the applied conditions in the graphics window, each type in a unique color. Note that this includes the selected design conditions, as well as the fixed conditions applied to zones that are not selected for deformation.

`adjoint/design-tool/design-change/`export-displacements

Exports the computed optimal displacements.

`adjoint/design-tool/design-change/`export-stl

Exports specified surfaces from 3D cases as an `.stl` file.

`adjoint/design-tool/design-change/`mesh-history/

Provides text commands that allow you to compare the current mesh to a reference mesh.

`adjoint/design-tool/design-change/mesh-history/`export-displacements

Exports the displacements of nodes in selected surfaces with respect to their earlier position in a reference mesh.

`adjoint/design-tool/design-change/mesh-history/`overlay-reference-mesh

Displays in the graphics window how selected surfaces of the mesh have changed compared to a reference mesh (reference mesh in green, modified mesh in white).

`adjoint/design-tool/design-change/mesh-history/`set-reference-mesh

Sets the current mesh as the reference mesh, which can then be compared to later modified versions of the mesh.

`adjoint/design-tool/design-change/`modify-mesh

Applies the computed optimal displacement to the mesh.

`adjoint/design-tool/design-change/`multiobjective-weights

Sets the weights for multiple free-form objectives.

`adjoint/design-tool/design-change/`preview-mesh

Displays the planned changes in the graphics window for select surfaces (undeformed in white, deformed in green).

`adjoint/design-tool/design-change/`print-expected-changes

Prints the expected changes.

adjoint/design-tool/design-change/revert-mesh

Reverts (rejects) the last mesh modification.

adjoint/design-tool/design-change/select-conditions

Selects which conditions to apply to the mesh deformation.

adjoint/design-tool/design-change/select-morphing-method

Selects the morphing method.

adjoint/design-tool/design-change/select-strict-conditions

Selects constraining and/or deformation conditions you want strictly enforced (that is, applied to all of the nodes of the associated zones).

adjoint/design-tool/design-change/select-zones

Selects which zones are allowed to deform.

adjoint/design-tool/design-change/settings

Specifies global deformation scale and settings.

adjoint/design-tool/design-change/write-expected-changes

Writes out the expected changes in the observables for the computed optimal design change.

adjoint/design-tool/design-conditions/

Creates, deletes, displays (with various options), renames, and/or sets conditions on the geometry deformation.

adjoint/design-tool/numerics

Adjusts numerics settings for computing the optimal displacement.

adjoint/design-tool/objectives/

Menu to configure observable objectives.

adjoint/design-tool/objectives/include-current?

Optionally includes the most-recently computed observable sensitivity in the multi-objective design computation.

adjoint/design-tool/objectives/manage/

Menu to import, export, and manage sensitivity data.

adjoint/design-tool/objectives/set

Configures the objectives for loaded observables.

`adjoint/design-tool/region-conditions/`

Menu to configure conditions on the deformation region

`adjoint/design-tool/region-conditions/boundaries`

Specifies what degree of continuity to enforce at the boundaries of the deformation region.

`adjoint/design-tool/region-conditions/invariant`

Specifies that the X, Y, and Z motion of the deformation region is uniform with respect to one or more of the coordinate directions. You have the option of enabling advanced settings if want greater control of this invariance (for example, if you only want the X motion to be uniform with respect to a particular direction).

`adjoint/design-tool/region-conditions/motion`

Enables/disables the motion of the control points in each of the coordinate directions.

`adjoint/design-tool/region-conditions/points`

Specifies the number of control points in the deformation region.

`adjoint/design-tool/region-conditions/symmetry`

Specifies symmetry conditions for the deformation region.

`adjoint/design-tool/region/`

Menu to define the deformation region.

`adjoint/design-tool/region/cartesian-limits`

Directly specifies the bounds of the Cartesian deformation region.

`adjoint/design-tool/region/cylindrical-limits`

Directly specifies the bounds of the cylindrical deformation region.

`adjoint/design-tool/region/export-sensitivities`

Exports the sensitivity to mesh node movement for the currently selected observable. Sensitivities are only exported for mesh nodes that lie within the specified deformation region.

`adjoint/design-tool/region/get-bounds`

Sets the limits for the deformation region to encompass a list of selected surfaces.

`adjoint/design-tool/region/larger-box`

Uniformly increases the size of the deformation region.

262

`adjoint/design-tool/region/`set-region-type

Specifies whether the deformation region is either a Cartesian or cylindrical region type.

`adjoint/design-tool/region/`smaller-box

Uniformly decreases the size of the deformation region.

`adjoint/`expert/

Provides text commands that allow you to undo enhancements to the default adjoint solver behavior.

`adjoint/expert/`undo-2019r3-default-changes?

Allows you to undo an enhancement introduced in version 2019 R3 of Ansys Fluent: the use of a full (rather than a flexible) cycle algebraic multigrid (AMG) solver in the adjoint solver, which can improve convergence and efficiency.

`adjoint/expert/`undo-r19.2-default-changes?

Allows you to undo enhancements introduced in version 19.2 of Ansys Fluent, including a modified Rhie-Chow averaging method (to be consistent with default changes introduced in the general solver in version 19.0), a non-conservative form of the adjoint equations, an enhanced method for building the algebraic multigrid (AMG) matrix for the adjoint solver, and a disabling of the early protection scheme used by the AMG solver for the adjoint calculations.

`adjoint/expert/`diagnosis

Provides text commands to enable printing/saving of adjoint equation residuals.

`adjoint/expert/diagnosis/`print-residuals-by-zone

Specifies that the residuals are printed in the console during the adjoint calculation.

`adjoint/expert/diagnosis/`retain-cell-residuals

Saves the residuals as field variables that can be postprocessed.

`adjoint/expert/diagnosis/`print-residuals-by-distribution

Specifies that the distribution of the residuals is printed in the console during the adjoint calculation.

`adjoint/`methods/

Menu to configure adjoint solver settings.

`adjoint/methods/`settings

Defines settings such as spatial discretization methods for the adjoint solver.

`adjoint/methods/`default-settings

Applies defaults to adjoint solver settings such as spatial discretization methods.

adjoint/methods/balanced-settings

Defines discretization schemes to achieve a balance between stability and accuracy.

adjoint/methods/best-match-settings

Defines settings such as spatial discretization methods for the adjoint solver to match those for the flow solver as closely as possible.

adjoint/monitors/

Menu to configure monitors for the adjoint solver.

adjoint/monitors/plot-residuals

Plots the adjoint residuals in the designated graphics window.

adjoint/monitors/settings

Allows you to configure the monitors and convergence criteria for the adjoint solver.

adjoint/observable/

Menu to create and configure observables of interest.

adjoint/observable/create

A new observable of the specified type and name is created and the definition is populated with default parameters.

adjoint/observable/delete

Removes a named observable.

adjoint/observable/evaluate

Evaluates the current value of the selected observable and prints the result to the console.

adjoint/observable/rename

An existing observable is renamed to a new specified name.

adjoint/observable/select

A named observable is selected as the one for which an adjoint solution is to be computed.

adjoint/observable/specify

The parameters that define a named observable are configured.

adjoint/observable/write

Evaluates the current value of the selected observable and writes the result to a file.

adjoint/optimizer/

Enters the optimizer menu, where you can use a gradient-based optimizer to optimize the geometry to meet multiple goals for multiple observables at multiple operating conditions.

adjoint/optimizer/calculate/

Enters the calculate menu, for running the gradient-based optimizer.

adjoint/optimizer/calculate/initialize

Initializes the optimizer and clears out the optimization history.

adjoint/optimizer/calculate/optimize

Runs adjoint-based optimization.

adjoint/optimizer/calculate/reset

Sets the current design iteration as the optimal solution.

adjoint/optimizer/calculate/summarize

Reports the optimization history.

adjoint/optimizer/calculation-activities/

Enters the calculation-activities menu, for setting activities during the adjoint optimization.

adjoint/optimizer/calculation-activities/animate/

Enters the animate menu, for outputting plots for animations every design iteration.

adjoint/optimizer/calculation-activities/animate/define/

Enters the animation definition menu.

adjoint/optimizer/calculation-activities/animate/define/define-monitor

Defines new animation.

adjoint/optimizer/calculation-activities/animate/define/edit-monitor

Change animation monitor attributes.

adjoint/optimizer/calculation-activities/animate/objects/

Enters the objects menu to define, edit, delete solution animation objects.

adjoint/optimizer/calculation-activities/animate/objects/clear-history

Clears object history.

adjoint/optimizer/calculation-activities/animate/objects/copy

Copies graphics object.

adjoint/optimizer/calculation-activities/animate/objects/create

Creates new graphics object.

adjoint/optimizer/calculation-activities/animate/objects/delete

Deletes graphics object.

adjoint/optimizer/calculation-activities/animate/objects/edit

Edits graphics object.

adjoint/optimizer/calculation-activities/animate/playback/

Enters the animation playback menu.

adjoint/optimizer/calculation-activities/animate/playback/delete

Deletes animation sequence.

adjoint/optimizer/calculation-activities/animate/playback/play

Plays the selected animation.

adjoint/optimizer/calculation-activities/animate/playback/read

Reads new animation from file or already-defined animations.

adjoint/optimizer/calculation-activities/animate/playback/stored-view?

Plays the 3D animation sequence using the view stored in the sequence.

adjoint/optimizer/calculation-activities/animate/playback/write

Writes animation sequence to the file.

adjoint/optimizer/calculation-activities/auto-save

Saves files at a specified frequency of design iterations.

adjoint/optimizer/calculation-activities/execute-commands/

Enters the execute-commands menu.

adjoint/optimizer/calculation-activities/execute-commands/add-edit

Adds or edits a command so that it can be enabled to be executed during the adjoint optimization.

`adjoint/optimizer/calculation-activities/execute-commands/disable`

Disables a command so that it is not executed during the adjoint optimization.

`adjoint/optimizer/calculation-activities/execute-commands/enable`

Enables a command so that it is executed during the adjoint optimization.

`adjoint/optimizer/calculation-activities/monitor`

Monitors the adjoint optimization.

`adjoint/optimizer/calculation-activities/monitor/plot`

Plots monitors.

`adjoint/optimizer/calculation-activities/monitor/settings`

Settings for monitors.

`adjoint/optimizer/mesh-quality/`

Enters the mesh-quality menu, for printing and setting the requirement of mesh quality during the adjoint optimization.

`adjoint/optimizer/mesh-quality/mesh-quality-settings`

Sets the requirement of mesh quality during the adjoint optimization.

`adjoint/optimizer/mesh-quality/print-current-status`

Prints the current mesh quality status.

`adjoint/optimizer/optimizer-objectives/`

Enters the optimizer-objectives menu, for setting the objectives in the adjoint optimizer.

`adjoint/optimizer/optimizer-objectives/set-adaptive-change`

Sets adaptive change.

`adjoint/optimizer/optimizer-objectives/set-conditions`

Sets operating conditions.

`adjoint/optimizer/optimizer-objectives/set-objectives`

Sets multi-objectives.

`adjoint/optimizer/optimizer-objectives/set-observables`

Choose observables.

`adjoint/optimizer/optimizer-settings`

Sets the settings in the adjoint optimizer.

`adjoint/`reporting/

Menu to report sensitivity data from the adjoint solution.

`adjoint/reporting/`report

Reports sensitivity data on a named flow boundary.

`adjoint/reporting/`write

Reports sensitivity data on a named flow boundary and writes it to a named file.

`adjoint/`run/

Menu to initialize and compute the adjoint solution.

`adjoint/run/`calculation-activities/

Enters the calculation-activities menu, for setting activities during the adjoint calculation.

`adjoint/run/calculation-activities/`auto-save

Saves case and data files at a specified frequency of adjoint iterations.

`adjoint/run/calculation-activities/`execute-commands/

Enters the execute-commands menu.

`adjoint/run/calculation-activities/execute-commands/`add-edit

Adds or edits a command that can be enabled to be executed during the adjoint calculation.

`adjoint/run/calculation-activities/execute-commands/`disable

Disables a command so that it is not executed during the adjoint calculation.

`adjoint/run/calculation-activities/execute-commands/`enable

Enables a command so that it is executed during the adjoint calculation.

`adjoint/run/`initialize

Initializes the adjoint solution field to zero everywhere.

`adjoint/run/`initialize-stabilization

Initializes the stabilization data. This text command is only available if the residual minimization or modal stabilization is used.

`adjoint/run/`initialize-strategy

Initializes the stabilization strategy, such that the first stabilization scheme is used at the start of the next calculation rather than continuing to use the second stabilization scheme. This text command is only available when the blended stabilization strategy is used.

268

`adjoint/run/iterate`

Advances the adjoint solver by a specified number of iterations, or until the convergence criteria are met.

Chapter 2: `define/`

`define/boundary-conditions/`

Enters 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 an asterisk (*) in the name of the zone to which you want to copy the conditions.

`define/boundary-conditions/axis`

Sets boundary conditions for a zone of this type.

`define/boundary-conditions/bc-settings/`

Enters the boundary conditions settings menu.

`define/boundary-conditions/bc-settings/mass-flow`

Selects method for setting the mass flow rate.

`define/boundary-conditions/bc-settings/pressure-outlet`

Sets advanced options for pressure outlet boundaries.

`define/boundary-conditions/bc-settings/pressure-far-field`

Sets advanced options for pressure far field boundaries.

`define/boundary-conditions/copy-bc`

Copies boundary conditions to other zones.

`define/boundary-conditions/degassing`

Sets boundary conditions for a zone of this type.

`define/boundary-conditions/exhaust-fan`

Sets boundary conditions for a zone of this type.

`define/boundary-conditions/fan`

Sets boundary conditions for a zone of this type.

define/boundary-conditions/fluid

Sets boundary conditions for a zone of this type.

define/boundary-conditions/impedance-data-fitting/

Enters the impedance data fitting menu, which provides text commands that can be helpful when you are using the impedance boundary condition (IBC).

define/boundary-conditions/impedance-data-fitting/absorption-data

Reads an input file with experimental absorption coefficient data in the frequency domain and then computes the terms needed for an approximation of the reflection coefficient as a series of poles / residues in the time domain. The resulting terms are printed in the console, and can be written to a pole / residue file (which can be imported and the impedance parameters applied to a boundary condition using the `define/boundary-conditions/impedance-data-fitting/import-parameters` text command). You can also write a file with fitted frequency / impedance data, which can be compared to the input data to evaluate how well it fits.

define/boundary-conditions/impedance-data-fitting/convergence-tolerance

Sets the convergence tolerance, which is an accuracy that is used for completing the iterative fitting procedure in the calculations performed from the `define/boundary-conditions/impedance-data-fitting/` text command menu. By default, the tolerance is set to 1e-6.

define/boundary-conditions/impedance-data-fitting/impedance-data

Reads an input file with experimental specific impedance data in the frequency domain and then computes the terms needed for an approximation of the reflection coefficient as a series of poles / residues in the time domain. The resulting terms are printed in the console, and can be written to a pole / residue file (which can be imported and the impedance parameters applied to a boundary condition using the `define/boundary-conditions/impedance-data-fitting/import-parameters` text command). You can also write a file with fitted frequency / impedance data, which can be compared to the input data to evaluate how well it fits.

define/boundary-conditions/impedance-data-fitting/import-parameters

Reads a pole / residue file with impedance parameters in the time domain and applies them to a specified boundary condition.

define/boundary-conditions/impedance-data-fitting/iterations

Sets the number of internal iterations used for the calculations performed from the `define/boundary-conditions/impedance-data-fitting/` text command menu. By default, the number of iterations is set to 20.

define/boundary-conditions/impedance-data-fitting/reflection-data

Reads an input file with experimental reflection coefficient data in the frequency domain and then computes the terms needed for an approximation of the reflection coefficient as a series of poles / residues in the time domain. The resulting terms are printed in the console,

and can be written to a pole / residue file (which can be imported and the impedance parameters applied to a boundary condition using the `define/boundary-conditions/impedance-data-fitting/import-parameters` text command). You can also write a file with fitted frequency / impedance data, which can be compared to the input data to evaluate how well it fits.

`define/boundary-conditions/impedance-data-fitting/residue-tolerance`

Sets the residue tolerance, which is the minimum value of residues that are kept in the fitting performed from the `define/boundary-conditions/impedance-data-fitting/` text command menu. This residue check helps to eliminate parasitic poles. By default, the tolerance is set to 1e-6.

`define/boundary-conditions/impedance-data-fitting/verbosity`

Sets the verbosity of the progress messages during the fitting performed from the `define/boundary-conditions/impedance-data-fitting/` text command menu. Setting this to `1` results in messages being printed in the console as the fitting calculation progresses. The default value of `0` suppresses this output.

`define/boundary-conditions/`**inlet-vent**

Sets boundary conditions for a zone of this type.

`define/boundary-conditions/`**intake-fan**

Sets boundary conditions for a zone of this type.

`define/boundary-conditions/`**interface**

Sets boundary conditions for a zone of this type.

`define/boundary-conditions/`**interior**

Sets boundary conditions for a zone of this type.

`define/boundary-conditions/`**knudsen-number-calculator**

Computes the Knudsen number based on a characteristic physical length and the area-averaged flow quantities along an incoming-flow boundary. You can use this information to determine flow regime for selecting the appropriate wall boundary treatment. For details, see Partial Slip for Rarefied Gases in the *Fluent User's Guide*.

`define/boundary-conditions/`**list-zones**

Prints 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.

`define/boundary-conditions/`**mass-flow-inlet**

Sets boundary conditions for a zone of this type.

define/boundary-conditions/mass-flow-outlet

Sets boundary conditions for a zone of this type.

define/boundary-conditions/modify-zones/

Enters the modify zones menu.

define/boundary-conditions/modify-zones/activate-cell-zone

Activates cell thread.

define/boundary-conditions/modify-zones/append-mesh

Appends new mesh.

define/boundary-conditions/modify-zones/append-mesh-data

Appends new mesh with data.

define/boundary-conditions/modify-zones/change-zone-state

Sets the state (liquid or vapor) for a specific fluid zone.

define/boundary-conditions/modify-zones/convert-all-solid-mrf-to-solid-motion

Converts all solid zones using frame motion to instead use solid motion, copying the motion variable values for origin, axis, and velocities.

define/boundary-conditions/modify-zones/copy-move-cell-zone

Creates 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*).

define/boundary-conditions/modify-zones/copy-mrf-to-mesh-motion

Copies motion variable values for origin, axis, and velocities from Frame Motion to Mesh Motion.

define/boundary-conditions/modify-zones/copy-mesh-to-mrf-motion

Copies motion variable values for origin, axis, and velocities from Mesh Motion to Frame Motion.

define/boundary-conditions/modify-zones/create-all-shell-threads

Marks all finite thickness walls for shell creation. Shell zones will be created at the start of the iterations.

define/boundary-conditions/modify-zones/create-periodic-interface

Creates a conformal or non-conformal periodic interface.

define/boundary-conditions/modify-zones/deactivate-cell-zone

Deactivates cell thread.

define/boundary-conditions/modify-zones/delete-all-shells

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

define/boundary-conditions/modify-zones/delete-cell-zone

Deletes a cell thread.

define/boundary-conditions/modify-zones/extrude-face-zone-delta

Extrudes a face thread a specified distance based on a list of deltas.

define/boundary-conditions/modify-zones/extrude-face-zone-para

Extrudes 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.

define/boundary-conditions/modify-zones/fuse-face-zones

Attempts to fuse zones by removing duplicate faces and nodes.

define/boundary-conditions/modify-zones/list-zones

Lists zone IDs, types, kinds, and names.

define/boundary-conditions/modify-zones/make-periodic

Attempts to establish periodic/shadow face zone connectivity.

define/boundary-conditions/modify-zones/matching-tolerance

Sets normalized tolerance used for finding coincident nodes.

define/boundary-conditions/modify-zones/merge-zones

Merges zones of same type and condition into one.

define/boundary-conditions/modify-zones/mrf-to-sliding-mesh

Changes the motion specification from MRF to moving mesh.

define/boundary-conditions/modify-zones/orient-face-zone

Orients the face zone.

define/boundary-conditions/modify-zones/recreate-all-shells

Recreates shells on all the walls that were deleted using the command `delete-all-shells`.

define/boundary-conditions/modify-zones/replace-zone

Replaces cell zone.

define/boundary-conditions/modify-zones/rotate-zone

Rotates individual cell zones.

define/boundary-conditions/modify-zones/scale-zone

Scales individual cell zones.

define/boundary-conditions/modify-zones/sep-cell-zone-mark

Separates cell zone based on cell marking.

define/boundary-conditions/modify-zones/sep-cell-zone-region

Separates cell zone based on contiguous regions.

define/boundary-conditions/modify-zones/sep-face-zone-angle

Separates face zone based on significant angle.

define/boundary-conditions/modify-zones/sep-face-zone-face

Separates each face in zone into unique zone.

define/boundary-conditions/modify-zones/sep-face-zone-mark

Separates face zone based on cell marking.

define/boundary-conditions/modify-zones/sep-face-zone-region

Separates face zone based on contiguous regions.

define/boundary-conditions/modify-zones/slit-periodic

Slits periodic zone into two symmetry zones.

define/boundary-conditions/modify-zones/slit-face-zone

Slits two-sided wall into two connected wall zones.

define/boundary-conditions/modify-zones/slit-interior-between-diff-solids

Slits 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.

define/boundary-conditions/modify-zones/translate-zone

Translates individual cell zones.

define/boundary-conditions/modify-zones/zone-name

Gives a zone a new name.

define/boundary-conditions/modify-zones/zone-type

Sets the type for a zone or multiple zones of the same category. You will be prompted for the name / ID of the zone to be changed and the new type for that zone. To change multiple zones, you can enter a list (separated by spaces and contained within a pair of parentheses) or use asterisks (*) as wildcards.

define/boundary-conditions/non-overlapping-zone-name

Displays the name of the non-overlapping zone associated with a specified interface zone. This text command is only available after a mesh interface has been created.

define/boundary-conditions/non-reflecting-bc/

Enters the non-reflecting boundary condition menu.

define/boundary-conditions/non-reflecting-bc/general-nrbc/

Setting for general non-reflecting b.c.

define/boundary-conditions/non-reflecting-bc/general-nrbc/set/

Enters the setup menu for general non-reflecting b.c.'s.

define/boundary-conditions/non-reflecting-bc/general-nrbc/set/sigma

Sets NRBC sigma factor (default value 0.15).

define/boundary-conditions/non-reflecting-bc/general-nrbc/set/sigma2

Sets NRBC sigma2 factor (default value 5.0).

define/boundary-conditions/non-reflecting-bc/general-nrbc/set/verbosity

Enables/disables nrbc verbosity scheme output.

define/boundary-conditions/non-reflecting-bc/turbo-specific-nrbc/

Enters the turbo specific nrbc menu.

define/boundary-conditions/non-reflecting-bc/turbo-specific-nrbc/enable?

Enables/disables non-reflecting b.c.'s.

define/boundary-conditions/non-reflecting-bc/turbo-specific-nrbc/initialize

Initializes non-reflecting b.c.'s.

define/boundary-conditions/non-reflecting-bc/turbo-specific-nrbc/set/

Enters the set menu for non-reflecting b.c. parameters.

define/boundary-conditions/non-reflecting-bc/turbo-specific-nrbc/set/discretization

Enables use of higher-order reconstruction at boundaries if available.

define/boundary-conditions/non-reflecting-bc/turbo-specific-nrbc/set/under-relaxation

Sets non-reflecting b.c. under-relaxation factor.

define/boundary-conditions/non-reflecting-bc/turbo-specific-nrbc/set/verbosity

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

define/boundary-conditions/non-reflecting-bc/turbo-specific-nrbc/show-status

Shows current status of non-reflecting b.c.'s.

define/boundary-conditions/open-channel-wave-settings

Opens channel wave input analysis.

define/boundary-conditions/openchannel-threads

Lists open channel group IDs, names, types and variables.

define/boundary-conditions/outflow

Sets boundary conditions for a zone of this type.

define/boundary-conditions/outlet-vent

Sets boundary conditions for a zone of this type.

define/boundary-conditions/perforated-walls/

Enters the perforated walls menu.

define/boundary-conditions/perforated-walls/model-setup

Sets up the perforated wall model for each effusion surface.

define/boundary-conditions/perforated-walls/read-input-file

Reads an input file for perforated walls.

define/boundary-conditions/periodic

Sets boundary conditions for a zone of this type.

define/boundary-conditions/phase-shift/

Enters the phase shift settings menu.

define/boundary-conditions/phase-shift/multi-disturbances

Sets basic phase-shift parameters.

define/boundary-conditions/phase-shift/extra-settings

Sets other phase-shift parameters.

define/boundary-conditions/porous-jump

Sets boundary conditions for a zone of this type.

define/boundary-conditions/pressure-far-field

Sets boundary conditions for a zone of this type.

define/boundary-conditions/pressure-inlet

Sets boundary conditions for a zone of this type.

define/boundary-conditions/pressure-outlet

Sets boundary conditions for a zone of this type.

define/boundary-conditions/radiator

Sets boundary conditions for a zone of this type.

define/boundary-conditions/rans-les-interface

Sets boundary conditions for a zone of this type.

define/boundary-conditions/recirculation-inlet

Sets boundary conditions for a zone of this type.

define/boundary-conditions/recirculation-outlet

Sets boundary conditions for a zone of this type.

define/boundary-conditions/rename-zone/

Enter the menu for renaming face zones based on adjacency, adding suffixes or prefixes to names, and renaming zones based on the Fluent naming conventions.

define/boundary-conditions/rename-zone/add-suffix-or-prefix

Add a suffix or prefix to one or more zones.

define/boundary-conditions/rename-zone/rename-by-adjacency

Rename a zone based on the name of the adjacent cell zone.

Appends the name of the adjacent cell zone to the specified face zone. For example, if **fluid** is the adjacent cell zone, and you select a face zone named **interior**, the resulting name of the face zone would be **interior-fluid**.

define/boundary-conditions/rename-zone/rename-to-default

Rename a zone to use the Fluent naming convention.

define/boundary-conditions/set/

Allows you to define one or more settings at single or multiple boundaries/cell zones of a given type at once. Enter **q** to exit the define/boundary-conditions/set/*<type>* command.

For a description of the items in this menu, see corresponding define/boundary-conditions/*<type>* .

define/boundary-conditions/shadow

Sets boundary conditions for a zone of this type.

define/boundary-conditions/solid

Sets boundary conditions for a zone of this type.

define/boundary-conditions/symmetry

Sets boundary conditions for a zone of this type.

define/boundary-conditions/target-mass-flow-rate-settings/

Enters the targeted mass flow rate settings menu.

define/boundary-conditions/target-mass-flow-rate-settings/setunder-relaxation-factor

The default setting is 0.05.

define/boundary-conditions/target-mass-flow-rate-settings/enabletargeted mass flow rate verbosity?

Enables/disables 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.

define/boundary-conditions/velocity-inlet

Sets boundary conditions for a zone of this type.

define/boundary-conditions/wall

Sets boundary conditions for a zone of this type.

define/boundary-conditions/zone-name

Gives a zone a new name.

define/boundary-conditions/zone-type

Sets the type for a zone or multiple zones of the same category. You will be prompted for the name / ID of the zone to be changed and the new type for that zone. To change multiple zones, you can enter a list (separated by spaces and contained within a pair of parentheses) or use asterisks (*) as wildcards.

define/custom-field-functions/

Enters the custom field functions menu.

define/custom-field-functions/define

Defines a custom field function.

define/custom-field-functions/delete

Deletes a custom field function.

define/custom-field-functions/example-cff-definitions

Lists example custom field functions.

define/custom-field-functions/list-valid-cell-function-names

Lists the names of cell functions that can be used in a custom field function.

define/custom-field-functions/load

Loads a custom field function.

define/custom-field-functions/save

Saves a custom field function.

define/dynamic-mesh/

Enters the dynamic mesh menu.

define/dynamic-mesh/actions/

Enters the dynamic mesh action menu, where you can initiate manual remeshing (that is, remeshing without running a calculation).

define/dynamic-mesh/actions/remesh-cell-zone

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

define/dynamic-mesh/controls/

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

define/dynamic-mesh/controls/contact-parameters/

Enters 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.

define/dynamic-mesh/controls/contact-parameters/contact-face-zones

Selects face zones involved in contact detection.

define/dynamic-mesh/controls/contact-parameters/contact-method

Selects the method used for flow control in the contact region. Enter `0` for the contact zones method (which restricts the flow using additional cell zones with porous zone properties) or `1` for the contact marks method (which blocks the flow using zero mass flux boundaries).

define/dynamic-mesh/controls/contact-parameters/contact-threshold

Specifies threshold distance for contact detection.

define/dynamic-mesh/controls/contact-parameters/contact-udf

Selects the UDF to be invoked when contact is detected.

define/dynamic-mesh/controls/contact-parameters/flow-control-parameters/

Enters the flow control parameters menu, which provides settings related to controlling the flow in the contact region.

define/dynamic-mesh/controls/contact-parameters/flow-control-parameters/create-flow-control-zone

Creates a flow control zone as part of the contact zones method.

define/dynamic-mesh/controls/contact-parameters/flow-control-parameters/delete-flow-control-zone

Deletes a flow control zone as part of the contact zones method.

define/dynamic-mesh/controls/contact-parameters/flow-control-parameters/solution-stabilization?

Enables/disables the performance of additional iterations per time step and the application of solution controls to improve the stability of the solver as part of the contact marks method.

define/dynamic-mesh/controls/contact-parameters/flow-control?

Enables/disables flow control in the contact region.

define/dynamic-mesh/controls/contact-parameters/render-contact-cells?

Enables/disables the availability of a field variable (**contact-cell-mark**) that can be used to display contours of cells marked for flow blocking as part of the contact marks method, and allows you to postprocess other field variables on those cells.

define/dynamic-mesh/controls/contact-parameters/update-contact-marks

Updates which cells are marked in order to block flow in the contact region as part of the contact marks method.

define/dynamic-mesh/controls/contact-parameters/verbosity

Sets the level of detail printed in the console regarding contact detection.

define/dynamic-mesh/controls/implicit-update-parameters/

Enters 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.

define/dynamic-mesh/controls/implicit-update-parameters/motion-relaxation

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

define/dynamic-mesh/controls/implicit-update-parameters/residual-criteria

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

define/dynamic-mesh/controls/implicit-update-parameters/update-interval

Specifies the update interval (that is, the frequency in iterations) at which the mesh is updated within a time step.

define/dynamic-mesh/controls/in-cylinder-output?

Enables/disables in-cylinder output.

define/dynamic-mesh/controls/in-cylinder-parameters/

Enters the dynamic mesh in-cylinder menu.

define/dynamic-mesh/controls/in-cylinder-parameters/crank-angle-step

Specifies crank angle step size.

define/dynamic-mesh/controls/in-cylinder-parameters/crank-period

Specifies the crank period.

define/dynamic-mesh/controls/in-cylinder-parameters/max-crank-angle-step

Specifies maximum crank angle step size.

define/dynamic-mesh/controls/in-cylinder-parameters/minimum-lift

Specifies minimum lift for in-cylinder valves.

define/dynamic-mesh/controls/in-cylinder-parameters/modify-lift

Modifies lift curve (shift or scale).

define/dynamic-mesh/controls/in-cylinder-parameters/piston-data

Specifies the crank radius and connecting rod length.

define/dynamic-mesh/controls/in-cylinder-parameters/piston-stroke-cutoff

Specifies the cut off point for in-cylinder piston.

define/dynamic-mesh/controls/in-cylinder-parameters/position-starting-mesh

Moves mesh from top dead center to starting crank angle.

define/dynamic-mesh/controls/in-cylinder-parameters/print-plot-lift

Prints or plot valve lift curve.

define/dynamic-mesh/controls/in-cylinder-parameters/starting-crank-angle

Specifies the initial value for the crank angle.

define/dynamic-mesh/controls/layering?

Enables/disables dynamic-layering in quad/hex cell zones.

define/dynamic-mesh/controls/layering-parameters/

Enters the dynamic mesh layering menu.

define/dynamic-mesh/controls/layering-parameters/collapse-factor

Sets the factor determining when to collapse dynamic layers.

define/dynamic-mesh/controls/layering-parameters/constant-height?

Enables/disables layering based on constant height, else layering based on constant ratio.

define/dynamic-mesh/controls/layering-parameters/split-factor

Sets the factor determining when to split dynamic layers.

define/dynamic-mesh/controls/periodic-displacement-parameters/

Enters the periodic displacement parameters menu.

define/dynamic-mesh/controls/periodic-displacement-parameters/create-properties

Sets the properties for periodic displacement.

`define/dynamic-mesh/controls/periodic-displacement-parameters/list-`
`properties`

Lists the properties for periodic displacement.

`define/dynamic-mesh/controls/remeshing?`

Enables/disables local remeshing in tri/tet and mixed cell zones.

`define/dynamic-mesh/controls/remeshing-parameters/`

Enters the dynamic mesh remeshing menu to set parameters for all remeshing methods.

`define/dynamic-mesh/controls/remeshing-parameters/cell-skew-max`

Sets the cell skewness threshold above which cells will be remeshed.

`define/dynamic-mesh/controls/remeshing-parameters/face-skew-max`

Sets the face skewness threshold above which faces will be remeshed.

`define/dynamic-mesh/controls/remeshing-parameters/length-max`

Sets the length threshold above which cells will be remeshed.

`define/dynamic-mesh/controls/remeshing-parameters/length-min`

Sets the length threshold below which cells will be remeshed.

`define/dynamic-mesh/controls/remeshing-parameters/parallel-remeshing?`

Disables/enables parallel remeshing as part of methods-based remeshing..

`define/dynamic-mesh/controls/remeshing-parameters/prism-layer-parameters/`

Enters the dynamic mesh prism layer parameters menu, where you can define the parameters of the prism layers as part of methods-based remeshing.

`define/dynamic-mesh/controls/remeshing-parameters/prism-layer-`
`parameters/first-height`

Sets the first cell height in the prism layer.

`define/dynamic-mesh/controls/remeshing-parameters/prism-layer-`
`parameters/growth-rate`

Sets the geometric growth rate of the prism layer.

`define/dynamic-mesh/controls/remeshing-parameters/prism-layer-`
`parameters/number-of-layers`

Sets the number of elements in the prism layer.

define/dynamic-mesh/controls/remeshing-parameters/remeshing-after-moving?

Enables a second round of remeshing based on the skewness parameters after the boundary has moved as part of methods-based remeshing.

define/dynamic-mesh/controls/remeshing-parameters/remeshing-methods

Enables/disables individual remeshing options as part of methods-based remeshing.

define/dynamic-mesh/controls/remeshing-parameters/retain-size-distribution?

Enables/disables the use of local size criteria when marking cells for unified remeshing (in an attempt to maintain the initial mesh size distribution even as the mesh moves), rather than marking cells based on the minimum and maximum length scale values of the cell zone in the initial mesh. Either marking can be overridden if more restrictive values are specified using the `define/dynamic-mesh/controls/remeshing-parameters/length-min` and `define/dynamic-mesh/controls/remeshing-parameters/length-max` text commands.

define/dynamic-mesh/controls/remeshing-parameters/size-remesh-interval

Sets the interval (in time steps) when remeshing based on size is done for methods-based remeshing.

define/dynamic-mesh/controls/remeshing-parameters/sizing-funct-defaults

Sets sizing function defaults.

define/dynamic-mesh/controls/remeshing-parameters/sizing-funct-rate

Determines how far from the boundary the increase/decrease happens.

define/dynamic-mesh/controls/remeshing-parameters/sizing-funct-resolution

Sets the sizing function resolution with respect to shortest boundary.

define/dynamic-mesh/controls/remeshing-parameters/sizing-funct-variation

Sets the maximum sizing function increase/decrease in the interior.

define/dynamic-mesh/controls/remeshing-parameters/sizing-function?

Enables/disables the sizing function as part of methods-based remeshing.

define/dynamic-mesh/controls/remeshing-parameters/unified-remeshing?

Enables/disables unified remeshing, which specifies that an algorithm is applied that combines aspects of a variety of remeshing methods to triangular or tetrahedral cells. Unified remeshing simplifies the remeshing setup and can provide increased robustness compared to methods-based remeshing, especially for parallel simulations.

define/dynamic-mesh/controls/remeshing-parameters/zone-remeshing

Enables/disables the cell zone remeshing method as part of methods-based remeshing.

`define/dynamic-mesh/controls/six-dof-parameters/`

Enters the dynamic mesh six degrees of freedom (DOF) solver menu.

`define/dynamic-mesh/controls/six-dof-parameters/create-properties`

Creates/edits a set of six DOF properties for rigid body motion.

`define/dynamic-mesh/controls/six-dof-parameters/delete-properties`

Deletes a set of six DOF properties for rigid body motion.

`define/dynamic-mesh/controls/six-dof-parameters/list-properties`

Prints summaries of the existing sets of six DOF properties for rigid body motion.

`define/dynamic-mesh/controls/six-dof-parameters/motion-history-file-name`

Specifies the name and location of the six DOF motion history file.

`define/dynamic-mesh/controls/six-dof-parameters/motion-history?`

Enables/disables writing position/orientation of six DOF zones to file.

`define/dynamic-mesh/controls/six-dof-parameters/second-order?`

Enables/disables the second order six degrees of freedom solver.

`define/dynamic-mesh/controls/six-dof-parameters/x-component-of-gravity`

Specifies x-component of gravity.

`define/dynamic-mesh/controls/six-dof-parameters/y-component-of-gravity`

Specifies y-component of gravity.

`define/dynamic-mesh/controls/six-dof-parameters/z-component-of-gravity`

Specifies z-component of gravity.

`define/dynamic-mesh/controls/smoothing?`

Enables/disables smoothing in cell zones.

`define/dynamic-mesh/controls/smoothing-parameters/`

Enters the dynamic mesh smoothing menu.

`define/dynamic-mesh/controls/smoothing-parameters/amg-stabilization`

Sets the algebraic multigrid (AMG) stabilization method for mesh smoothing calculations that use finite element discretization.

define/dynamic-mesh/controls/smoothing-parameters/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.

define/dynamic-mesh/controls/smoothing-parameters/bnd-stiffness-factor

Sets the stiffness factor for springs connected to boundary nodes.

define/dynamic-mesh/controls/smoothing-parameters/boundary-distance-method

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

define/dynamic-mesh/controls/smoothing-parameters/constant-factor

Sets the spring constant relaxation factor.

define/dynamic-mesh/controls/smoothing-parameters/convergence-tolerance

Sets the convergence tolerance for spring-based solver.

define/dynamic-mesh/controls/smoothing-parameters/diffusion-coeff-function

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

define/dynamic-mesh/controls/smoothing-parameters/diffusion-coeff-parameter

Sets the diffusion coefficient parameter used for diffusion-based smoothing.

define/dynamic-mesh/controls/smoothing-parameters/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.

define/dynamic-mesh/controls/smoothing-parameters/max-iter

Sets the maximum number of iterations for the smoothing calculations.

define/dynamic-mesh/controls/smoothing-parameters/relative-convergence-tolerance

Sets the relative residual convergence tolerance for smoothing based on diffusion or the linearly elastic solid model.

define/dynamic-mesh/controls/smoothing-parameters/skew-smooth-cell-skew-max

Sets the skewness threshold, above which cells will be smoothed using the skewness method.

define/dynamic-mesh/controls/smoothing-parameters/skew-smooth-face-skew-max

Sets the skewness threshold, above which faces will be smoothed using the skewness method.

define/dynamic-mesh/controls/smoothing-parameters/skew-smooth-niter

Sets the number of skewness-based smoothing cycles.

define/dynamic-mesh/controls/smoothing-parameters/smooth-from-reference-position?

Enables/disables smoothing from a reference position. Such smoothing may produce greater mesh quality consistency for stationary or moving meshes with periodic or quasi-periodic motion, and is only available when the smoothing method is based on diffusion or the linearly elastic solid model.

define/dynamic-mesh/controls/smoothing-parameters/smoothing-method

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

define/dynamic-mesh/controls/smoothing-parameters/spring-on-all-elements?

Enables/disables spring-based smoothing for all cell shapes; if disabled, the spring-based smoothing is applied based on the setting of the `define/dynamic-mesh/controls/smoothing-parameters/spring-on-simplex-shapes?` text command.

define/dynamic-mesh/controls/smoothing-parameters/spring-on-simplex-shapes?

Enables/disables spring-based smoothing for triangular / tetrahedral cells in mixed element zones; if disabled, the spring-based smoothing is applied to triangular / tetrahedral cells in triangular / tetrahedral element zones.

define/dynamic-mesh/controls/smoothing-parameters/poisson-ratio

Sets the Poisson's ratio used for smoothing based on the linearly elastic solid model.

define/dynamic-mesh/controls/smoothing-parameters/verbosity

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

define/dynamic-mesh/controls/steady-pseudo-time-control?

Enables/disables the pseudo time step control in the graphical user interface.

`define/dynamic-mesh/`dynamic-mesh?

Enables/disables the dynamic mesh solver.

`define/dynamic-mesh/`events/

Enters the dynamic mesh events menu.

`define/dynamic-mesh/events/`export-event-file

Exports dynamic mesh events to file.

`define/dynamic-mesh/events/`import-event-file

Imports dynamic mesh event file.

`define/dynamic-mesh/`transient-settings/

Enters 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**.

`define/dynamic-mesh/transient-settings/`allow-second-order?

Enables/disables second order transient scheme for dynamic mesh cases.

`define/dynamic-mesh/transient-settings/`verbosity

Enables/disables transient scheme verbosity for dynamic mesh cases.

`define/dynamic-mesh/`zones/

Enters the dynamic mesh zones menu.

`define/dynamic-mesh/zones/`create

Creates or edit a dynamic zone.

`define/dynamic-mesh/zones/`delete

Deletes a dynamic zone.

`define/dynamic-mesh/zones/`insert-boundary-layer

Inserts a new cell zone.

`define/dynamic-mesh/zones/`insert-interior-layer

Inserts a new layer cell zone at a specified location.

`define/dynamic-mesh/zones/`list

Lists the dynamic zones.

define/dynamic-mesh/zones/remove-boundary-layer

Removes a cell zone.

define/dynamic-mesh/zones/remove-interior-layer

Removes an interior layer cell zone.

define/**enable-mesh-morpher-optimizer?**

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

define/**gap-model/**

Enters the gap model menu, where you can define one or more gap regions where the flow is controlled when face zones move within a specified proximity threshold of each other.

define/gap-model/**advanced-options/**

Enters the advanced options menu for the gap model.

define/gap-model/advanced-options/**check-cfl-condition?**

Enables/disables the printing of warnings if the time step size is too large based on a Courant (CFL) number automatically determined for your specified solution stability level.

define/gap-model/advanced-options/**enhanced-data-interpolation?**

Enables/disables the use of enhanced data interpolation when updating information in gap regions.

define/gap-model/advanced-options/**revert-controls-to-default**

Reverts the gap stabilization level and any related solver settings to the default.

define/gap-model/advanced-options/**solution-stabilization**

Sets the solution stabilization level for the gap model.

define/gap-model/advanced-options/**verbosity**

Sets the verbosity for messages printed in the console related to the gap model.

define/gap-model/**create**

Creates a single gap region, so that when selected face zones move within a specified proximity threshold of each other, flow control is applied to the cells that lie within the threshold.

define/gap-model/**delete**

Deletes an existing gap region.

define/gap-model/delete-all

Deletes all of the existing gap regions.

define/gap-model/edit

Edits an existing gap region.

define/gap-model/enable?

Enables/disables the gap model.

define/gap-model/list-gap-cell-zones

Lists the names of the cell zones that can be excluded for individual gap regions (so that such cells are not marked for flow blockage).

define/gap-model/list-gap-face-zones

Lists the names of the face zones that can be used for creating gap regions.

define/gap-model/list-gap-regions

Lists the properties of the gap regions.

define/injections/

Enters the injections menu.

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

define/materials/

Enters the materials menu.

define/materials/change-create

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

> **Note:**
>
> Generally, the properties you enter at the prompts will be filtered according to your case and model settings. However, some of the listed properties may not match the selection choice in the Graphics User Interface. Typically, those excessive properties in the Text User Interface will not be used in your simulation.

define/materials/copy

Copies a material from the database.

define/materials/copy-by-formula

Copies a material from the database by formula.

define/materials/data-base/

Enters the material database menu.

define/materials/data-base/database-type

Sets the database type.

define/materials/data-base/edit

Edits material.

define/materials/data-base/list-materials

Lists all materials in the database.

define/materials/data-base/list-properties

Lists the properties of a material in the database.

define/materials/data-base/new

Defines new material.

define/materials/data-base/save

Saves user-defined database.

define/materials/delete

Deletes a material from local storage.

define/materials/list-materials

Lists all locally-stored materials.

define/materials/list-properties

Lists the properties of a locally-stored material.

define/mesh-interfaces/

Enters the mesh-interfaces menu.

define/mesh-interfaces/auto-options/

Enters the auto-options menu. This menu is only available when the define/mesh-interfaces/one-to-one-pairing? text command is enabled.

define/mesh-interfaces/auto-options/naming-option

Specifies whether the name of each new one-to-one mesh interface (as well as existing mesh interfaces, if you so desire) has no additional suffix, or a suffix that includes the names of the associated boundary zones, the IDs of the associated boundary zones, or the names of the adjacent cell zones.

define/mesh-interfaces/auto-options/pairing-between-different-cell-zones-only?

Specifies whether the one-to-one mesh interfaces are only created between different cell zones.

define/mesh-interfaces/auto-options/proximity-tolerance

Sets the tolerance used when automatically grouping zones to create one-to-one mesh interfaces. The proximity tolerance is defined relative to the edge lengths in the interface zones, and can range from 0 to 1 (representing the minimum and maximum edge lengths, respectively).

define/mesh-interfaces/auto-options/set-default-name-prefix

Specifies the default interface name prefix used for one-to-one mesh interfaces.

define/mesh-interfaces/auto-pairing

Automatically "pairs" some or all of the interface zones, in order to create mesh interfaces. This text command is only available when the `define/mesh-interfaces/one-to-one-pairing?` text command is disabled.

define/mesh-interfaces/create

Creates mesh interfaces.

define/mesh-interfaces/delete

Deletes a mesh interface.

define/mesh-interfaces/delete-all

Deletes all mesh interfaces.

define/mesh-interfaces/display

Displays the specified mesh interface zone.

define/mesh-interfaces/edit

Edits attributes of existing mesh interfaces. For one-to-one interfaces, you can edit the name; for many-to-many interfaces, you can edit the interface options and (for a single interface) the name and the list of interface zones assigned to the interface.

define/mesh-interfaces/enable-visualization-of-interfaces?

Enables/disables the filling of node coordinates on the zones of mesh interfaces, so that they can be displayed in the graphics window.

define/mesh-interfaces/enforce-continuity-after-bc?

Enables/disables continuity across the boundary condition interface for contour plots in post-processing.

define/mesh-interfaces/enforce-coupled-wall-between-solids?

Enables/disables automatic definition of solid-solid interfaces as coupled walls. By default this option is disabled and Ansys Fluent creates interior boundaries at solid-solid interfaces.

define/mesh-interfaces/improve-quality

Checks the quality of all mapped interfaces. If Fluent finds any mapped interfaces that require improvement it will list them and ask you if you would like to increase the tolerance to improve the interfaces.

define/mesh-interfaces/list

Lists all mesh interfaces.

define/mesh-interfaces/mapped-interface-options/

Enters the mapped-interface menu. Options are available to convert a non-conformal interface to a mapped interface. This menu is only available when the `define/mesh-interfaces/one-to-one-pairing?` text command is disabled.

define/mesh-interfaces/mapped-interface-options/convert-to-mapped-interface

Converts non-conformal mesh interface to mapped mesh interfaces. Answering `yes` to `Convert all mesh interfaces to mapped mesh interfaces?` will enforce the mapping such that it specifies all interfaces as mapped interfaces; the mapped interfaces are created regardless of mesh quality. Answering `yes` to `Convert poorly matching mesh interfaces to mapped mesh interfaces?` will automate the mapping such that all interfaces that penetrate each other are automatically defined as mapped interfaces. Note that in all cases, only interfaces for which at least one side of the interface consists of only solid zones will be converted.

define/mesh-interfaces/mapped-interface-options/solution-controls

Specifies the mapped frequency and under-relaxation factor for mapped interfaces.

define/mesh-interfaces/mapped-interface-options/tolerance

Specifies the mapped interface tolerance. After changing the tolerance, the existing mapped interface will be updated.

define/mesh-interfaces/non-conformal-interface-numerics/

Enters the non-conformal-interface-numerics menu.

define/mesh-interfaces/non-conformal-interface-numerics/change-numerics?

Provides options to improve numerics for non-conformal interfaces in some cases.

- `Enforce watertight cells for fluid-solid and solid-solid interfaces?`: Answering `yes` ensures cells adjacent to a non-conformal fluid-solid or solid-solid interface are watertight when performing the coupled wall discretization.

- Use enhanced gradient computations for fluid-solid and solid-solid interfaces?: Answering `yes` enables improvements to the calculation of gradients.

- `Recreate non-conformal interfaces?`: If you change the response to either of the previous two questions then interfaces should be recreated.

define/mesh-interfaces/**one-to-one-pairing?**

Enables/disables the use of the one-to-one interface creation method. This method is enabled by default, such that a single zone is assigned to each side of all of the mesh interfaces when they are created; otherwise the mesh interfaces may have multiple zones on one or both sides. Any mesh interfaces that exist when the use of this method is changed will either be deleted or modified to be compatible with this method.

define/mesh-interfaces/**remove-left-handed-interface-faces?**

Removes left-handed faces (which can cause the mesh to be invalid) during mesh interface creation.

define/mesh-interfaces/**transfer-motion-across-interfaces?**

Enables/disables the automatic transfer of motion across a mesh interface when only one side is moving as a result of user-defined or system coupling motion. You can specify the method by which the motion is transferred: `transfer-displacements` (the default) interpolates nodal displacement from the active side of the interface to the passive side, and is recommended when there are gaps and/or penetrations in the mesh interface that must be maintained; `project-nodes` projects the passive nodes onto the faces of active side, and is recommended when the active side includes significant tangential motion (as only the normal displacement is effectively transferred in this method).

define/mesh-interfaces/**turbo-create**

Creates a general turbo interface. This text command is only available if the `define/turbo-model/enable-turbo-model?` text command is enabled.

define/mesh-interfaces/**use-virtual-polygon-approach**

Uses 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.

define/mesh-interfaces/**verbosity**

Sets the mesh interface verbosity.

define/mesh-morpher-optimizer/

Enters 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.

define/mesh-morpher-optimizer/deformation-settings/

Enters 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.

define/mesh-morpher-optimizer/deformation-settings/check-mesh

Displays 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.

define/mesh-morpher-optimizer/deformation-settings/deform-mesh

Modifies the mesh and updates 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.

define/mesh-morpher-optimizer/deformation-settings/motion-settings/

Enters the motion settings menu, where you can create, modify, and delete the motions applied to the control points. This text command is only available if the `define/mesh-morpher-optimizer/set-control-point-distribution` text command is set to unstructured.

define/mesh-morpher-optimizer/deformation-settings/motion-settings/create

Creates a new motion, by specifying the parameter, type of motion (translation, rotation, or radial motion), directional settings, name, and control points. Note that for translations, the values you enter for the direction components are multiplied with the value of the parameter to define the displacement applied to the control points.

define/mesh-morpher-optimizer/deformation-settings/motion-settings/delete

Deletes an existing motion.

define/mesh-morpher-optimizer/deformation-settings/motion-settings/modify

Modifies the settings for an existing motion.

define/mesh-morpher-optimizer/deformation-settings/motion-settings/read-from-file

Defines the settings for all the motions by reading an ASCII text file.

define/mesh-morpher-optimizer/deformation-settings/motion-settings/write-to-file

Writes the settings for all the motions to an ASCII text file.

define/mesh-morpher-optimizer/deformation-settings/read-motion-settings-from-file

Defines all of the motion settings by reading an ASCII text file. This text command is only available if the `define/mesh-morpher-optimizer/set-control-point-distribution` text command is set to regular.

define/mesh-morpher-optimizer/deformation-settings/reset-all-deformations

Undoes any deformations made to the mesh and updates the mesh display in the graphics window.

define/mesh-morpher-optimizer/deformation-settings/set-constraints

Defines the constraints on a boundary zone, in order to limit the freedom of a particular zone that falls within the deformation region(s) during the morphing of the mesh.

define/mesh-morpher-optimizer/deformation-settings/set-constraints-multiple

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

define/mesh-morpher-optimizer/deformation-settings/set-parameters

Assigns translation direction components and a single parameter to a single control point in a region. Note that the values you enter for the direction components are multiplied with the value of the parameter to define the displacement applied to the control point. This text command is only available if the `define/mesh-morpher-optimizer/set-control-point-distribution` text command is set to regular.

define/mesh-morpher-optimizer/deformation-settings/set-parameters-multiple

Assigns translation direction components and multiple parameters to multiple control points in a region. Note that the values you enter for the direction components are multiplied with the value of the parameter to define the displacement applied to the control point. This text command is only available if the `define/mesh-morpher-optimizer/set-control-point-distribution` text command is set to regular.

define/mesh-morpher-optimizer/deformation-settings/write-motion-settings-to-file

Writes the motion settings to an ASCII text file. This text command is only available if the `define/mesh-morpher-optimizer/set-control-point-distribution` text command is set to regular.

define/mesh-morpher-optimizer/morpher-summary

Displays 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.

define/mesh-morpher-optimizer/optimizer-parameters/

Enters 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.

define/mesh-morpher-optimizer/optimizer-parameters/convergence-criteria

Defines the convergence criteria for the optimizer.

define/mesh-morpher-optimizer/optimizer-parameters/custom-calculator/

Enters 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/mesh-morpher-optimizer/optimizer-parameters/custom-calculator/define

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

define/mesh-morpher-optimizer/optimizer-parameters/custom-calculator/delete

Deletes the saved custom objective function.

define/mesh-morpher-optimizer/optimizer-parameters/custom-calculator/example-obj-fn-definitions

Prints examples of custom objective function definitions in the console.

define/mesh-morpher-optimizer/optimizer-parameters/custom-calculator/list-output-parameters

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

define/mesh-morpher-optimizer/optimizer-parameters/disable-mesh-check

Specifies 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.

define/mesh-morpher-optimizer/optimizer-parameters/end-commands

Specifies 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.

define/mesh-morpher-optimizer/optimizer-parameters/initial-commands

Specifies 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.

define/mesh-morpher-optimizer/optimizer-parameters/initialization

Specifies 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.

define/mesh-morpher-optimizer/optimizer-parameters/iterations-per-design

Defines the maximum number of iterations Ansys Fluent will perform for each design change.

define/mesh-morpher-optimizer/optimizer-parameters/maximum-designs

Defines the maximum number of design stages the optimizer will undergo to reach the specified objective function.

define/mesh-morpher-optimizer/optimizer-parameters/mesh-quality-check

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

define/mesh-morpher-optimizer/optimizer-parameters/monitor/

Enters 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.

define/mesh-morpher-optimizer/optimizer-parameters/monitor/clear-opt-hist

Discards the optimization history data, including the associated files.

define/mesh-morpher-optimizer/optimizer-parameters/monitor/plot-hist

Displays 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-optimiza-
tion-monitor-data text command.

define/mesh-morpher-optimizer/optimizer-parameters/monitor/plot?

Enables the plotting of the optimization history data in the graphics window.

define/mesh-morpher-optimizer/optimizer-parameters/monitor/write?

Enables the saving of the optimization history data to a file.

define/mesh-morpher-optimizer/optimizer-parameters/newuoa-initial-parameter-variation

Defines 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.

define/mesh-morpher-optimizer/optimizer-parameters/objective-function-definition

Specifies 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.

define/mesh-morpher-optimizer/optimizer-parameters/optimize

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

define/mesh-morpher-optimizer/optimizer-parameters/optimizer-type

Specifies 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 Design Analysis and Optimization in the User's Guide or Working With Input and Output
Parameters in Workbench in the Ansys Fluent in Workbench User's Guide, respectively.

define/mesh-morpher-optimizer/optimizer-parameters/save-case-data-files

Sets 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).

define/mesh-morpher-optimizer/optimizer?

Enables 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.

define/mesh-morpher-optimizer/parameter-settings/

Enters the parameters menu.

define/mesh-morpher-optimizer/parameter-settings/number-of-parameters

Defines the number of parameters that will be used to define the deformation.

define/mesh-morpher-optimizer/parameter-settings/parameter-bounds

Defines 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.

define/mesh-morpher-optimizer/parameter-settings/parameter-value

Defines the value of a parameter. This value represents a magnitude that will then be used along with other direction settings to define motions that produce an overall displacement for a given control point. The value will specify a length in meters for translations and radial motions, and will specify an angle in degrees for rotations; the units here will be used regardless of what units you are using in the case. 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

define/mesh-morpher-optimizer/region/

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

define/mesh-morpher-optimizer/region/controlpoints/

Enters the control points menu in order to create, modify, and/or delete control points. This text command is only available if the `define/mesh-morpher-optimizer/set-control-point-distribution` text command is set to unstructured.

define/mesh-morpher-optimizer/region/controlpoints/create

Creates a new control point by defining the coordinates of a point within an existing deformation region.

define/mesh-morpher-optimizer/region/controlpoints/delete

Deletes existing control points.

define/mesh-morpher-optimizer/region/controlpoints/distribute-points-on-zone

Creates an approximate number of control points on the mesh nodes of a particular boundary zone, with a distribution that is based on the distribution of the cell faces in that zone.

define/mesh-morpher-optimizer/region/controlpoints/modify

Modifies the coordinates of existing control points.

define/mesh-morpher-optimizer/region/controlpoints/read-from-file

Defines all of the control points by reading an ASCII text file.

define/mesh-morpher-optimizer/region/controlpoints/write-to-file

Writes the control point coordinates to an ASCII text file.

define/mesh-morpher-optimizer/region/create

Creates a new deformation region by specifying the name, number of control points (for a regular distribution), dimensions, origin coordinates, the components of the direction vectors, and whether to smooth the transitions of the mesh at the region boundaries (for an unstructured distribution). 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/mesh-morpher-optimizer/region/define-bounding-box

Creates a new deformation region by specifying the name, the bounding zones that best represent the extents of the deformation region, the number of control points (for a regular distribution), and whether to smooth the transitions of the mesh at the region boundaries (for an unstructured distribution). 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/mesh-morpher-optimizer/region/delete

Deletes a deformation region.

define/mesh-morpher-optimizer/region/scaling-enlarge

Sets 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.

define/mesh-morpher-optimizer/region/scaling-reduce

Sets 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.

`define/mesh-morpher-optimizer/set-control-point-distribution`

Specifies whether the control point distribution is regular (that is, spread in a regular distribution throughout the entire deformation region) or unstructured (that is, distributed at specified locations).

`define/mixing-planes/`

Enters the mixing planes menu.

`define/mixing-planes/create`

Creates a mixing plane.

`define/mixing-planes/delete`

Deletes a mixing plane.

`define/mixing-planes/list`

Lists defined mixing plane(s).

`define/mixing-planes/set/`

Sets global parameters relevant to mixing planes.

`define/mixing-planes/set/averaging-method`

Sets the mixing plane profile averaging method.

`define/mixing-planes/set/under-relaxation`

Sets mixing plane under-relaxation factor.

`define/mixing-planes/set/fix-pressure-level`

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

`define/mixing-planes/set/conserve-swirl/`

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

`define/mixing-planes/set/conserve-swirl/enable?`

Enables/disables swirl conservation in mixing plane.

`define/mixing-planes/set/conserve-swirl/verbosity?`

Enables/disables verbosity in swirl conservation calculations.

`define/mixing-planes/set/conserve-swirl/report-swirl-integration`

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

define/mixing-planes/set/conserve-total-enthalpy/

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

define/mixing-planes/set/conserve-total-enthalpy/enable?

Enables/disables total enthalpy conservation in mixing plane.

define/mixing-planes/set/conserve-total-enthalpy/verbosity?

Enables/disables verbosity in total-enthalpy conservation calculations.

define/models/

Enters the models menu to configure the solver.

define/models/ablation?

Enables/disables the ablation model.

define/models/acoustics/

Enters the acoustics menu.

define/models/acoustics/auto-prune

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

define/models/acoustics/broad-band-noise?

Enables/disables the broadband noise model.

define/models/acoustics/convective-effects?

Enables/disables the convective effects option.

define/models/acoustics/compute-write

Computes sound pressure.

define/models/acoustics/cylindrical-export?

Enables/disables the export of data in cylindrical coordinates.

define/models/acoustics/display-flow-time?

Enables/disables the display of flow time during read-and-compute.

define/models/acoustics/export-source-data-cgns?

Enables/disables the export of acoustic source data in CGNS format.

define/models/acoustics/export-volumetric-sources?

Enables/disables the export of fluid zones.

define/models/acoustics/export-volumetric-sources-cgns?

Enables/disables the export of fluid zones.

define/models/acoustics/export-source-data?

Enables/disables the export of acoustic source data in ASD format during the wave equation model run.

define/models/acoustics/far-field-parameters/

Enters the menu to specify the far-field density and speed of sound. Note that this menu is currently available only with the acoustics wave equation model.

define/models/acoustics/far-field-parameters/far-field-density

Specifies the far-field density value for the acoustics wave equation model.

define/models/acoustics/far-field-parameters/far-field-sound-speed

Specifies the far-field speed of sound value for the acoustics wave equation model.

define/models/acoustics/ffowcs-williams?

Enables/disables the Ffowcs-Williams-and-Hawkings model.

define/models/acoustics/moving-receiver?

Enables/disables the moving receiver option.

define/models/acoustics/off?

Enables/disables the acoustics model.

define/models/acoustics/read-compute-write

Reads acoustic source data files and computes sound pressure.

define/models/acoustics/receivers

Sets acoustic receivers.

define/models/acoustics/sources

Sets acoustic sources.

define/models/acoustics/sources-fft/

Enters the acoustic sources fast Fourier transform (FFT) menu, to compute Fourier spectra from acoustic source data (ASD) files, create postprocessing variables for the pressure signals, and write CGNS files of the spectrum data.

define/models/acoustics/sources-fft/read-asd-files

Reads ASD files to perform FFT of the pressure history field.

define/models/acoustics/sources-fft/compute-fft-fields

Computes FFT of the read pressure histories. The computed spectra replace the pressure histories in memory.

define/models/acoustics/sources-fft/clean-up-storage-area

De-allocates memory used to store the pressure histories and their Fourier spectra, as well as any created surface variables for the visualization.

define/models/acoustics/sources-fft/write-cgns-files

Writes surface pressure spectra in CGNS format, which can be used for one-way coupling with Ansys Mechanical in the frequency domain.

define/models/acoustics/sources-fft/fft-surface-variables/

Enters the menu to create surface variables from the computed Fourier spectra for visualization.

define/models/acoustics/sources-fft/fft-surface-variables/create-constant-width-bands

Selects up to 20 constant width bands and creates either the surface pressures level (SPL) variables or the PSD of dp/dt variables for them.

define/models/acoustics/sources-fft/fft-surface-variables/create-octave-bands

Creates either the surface pressure level (SPL) variables or the PSD of dp/dt variables for 17 technical octaves.

define/models/acoustics/sources-fft/fft-surface-variables/create-third-bands

Creates either the surface pressure level (SPL) variables or the PSD of dp/dt variables for 54 technical thirds.

define/models/acoustics/sources-fft/fft-surface-variables/create-set-of-modes

Selects up to 20 individual Fourier modes and create variable pairs for them, containing the real and the imaginary parts of the complex Fourier amplitudes.

define/models/acoustics/sources-fft/fft-surface-variables/remove-variables

Removes all variables created in this menu.

define/models/acoustics/wave-equation?

Enables/disables the wave equation model.

define/models/acoustics/wave-equation-options/

Enters the menu to define the acoustics wave equation model options.

define/models/acoustics/wave-equation-options/basic-shapes/

Enters the menu to define the geometry of the source mask and sponge layer using the basic shapes, represented by the cell registers of the type "Region".

define/models/acoustics/wave-equation-options/basic-shapes/add-source-mask-shape

Adds a basic shape to the definition of the source mask geometry.

define/models/acoustics/wave-equation-options/basic-shapes/add-sponge-layer-shape

Adds a basic shape to the definition of the sponge layer geometry.

define/models/acoustics/wave-equation-options/basic-shapes/list-source-mask-shapes

List basic shapes, which are currently used in the definition of the source mask geometry.

define/models/acoustics/wave-equation-options/basic-shapes/list-sponge-layer-shapes

List basic shapes, which are currently used in the definition of the sponge layer geometry.

define/models/acoustics/wave-equation-options/basic-shapes/list-region-registers

List all available cell registers of the type "Region".

define/models/acoustics/wave-equation-options/basic-shapes/remove-source-mask-shape

Remove a basic shape from the definition of the source mask geometry.

define/models/acoustics/wave-equation-options/basic-shapes/remove-sponge-layer-shape

Remove a basic shape from the definition of the sponge layer geometry.

define/models/acoustics/wave-equation-options/remote-receivers-options/

Enters the menu to define remote receivers for the Kirchhoff integral model.

define/models/acoustics/wave-equation-options/remote-receivers-options/integration-surface

Selects the integration surface for the Kirchhoff model.

define/models/acoustics/wave-equation-options/remote-receivers-options/write-signals

Writes the computed receiver signals to the ASCII files.

define/models/acoustics/wave-equation-options/remote-receivers?

Enables/disables the Kirchhoff integral model.

define/models/acoustics/wave-equation-options/source-mask-udf

Specifies the name of a user-defined function, which defines geometry of the source mask.

define/models/acoustics/wave-equation-options/sponge-layer-base-value

Specifies the base value of the artificial viscosity coefficient.

define/models/acoustics/wave-equation-options/sponge-layer-factor

Specifies the factor of the artificial viscosity coefficient.

define/models/acoustics/wave-equation-options/sponge-layer-udf

Specifies the name of a user-defined function, which defines geometry of the sponge layer.

define/models/acoustics/wave-equation-options/time-filter-source?

Enables/disables a time filter for the sound source.

define/models/acoustics/write-acoustic-signals

Writes on-the-fly sound pressure.

define/models/acoustics/write-centroid-info

Writes centroid info.

define/models/addon-module

Loads addon module.

define/models/axisymmetric?

Specifies whether or not the domain is axisymmetric.

define/models/battery-model

Enables the dual potential MSMD battery model.

define/models/cht/

Enters the cht (conjugate heat transfer) menu.

define/models/cht/implicit-coupling?

Enables the implicit mapping scheme for any fluid-solid pair with a mapped mesh interface (only required for cases set up in version 19.2 or earlier).

define/models/cht/explicit-time-averaged-coupling

Enters the explicit time averaged thermal coupling menu.

conformal-coupled-walls

Specify explicit coupling controls for coupled conformal walls. Note this operation will slit the coupled wall pair.

coupling-controls

Specifies the number of time steps per coupling iteration.

fuse-explicit-cht-zones

Fuses slitted conformal coupled walls marked for transient explicit thermal coupling.

define/models/crevice-model?

Enables/disables the crevice model.

define/models/crevice-model-controls/

Enters the crevice model controls menu.

define/models/dpm/

Enters the dispersed phase model menu.

define/models/dpm/clear-particles-from-domain

Removes/keeps all particles currently in the domain.

define/models/dpm/collisions/

Enters the DEM collisions menu.

define/models/dpm/collisions/collision-mesh

Input for the collision mesh.

define/models/dpm/collisions/collision-pair-settings/

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

define/models/dpm/collisions/collision-pair-settings/contact-force-normal

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

define/models/dpm/collisions/collision-pair-settings/contact-force-tangential

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

define/models/dpm/collisions/collision-pair-settings/list-pair-settings

Lists the current settings for this pair of collision partners.

define/models/dpm/collisions/collision-partners/

Manages collision partners.

define/models/dpm/collisions/collision-partners/copy

Copies a collision partner.

define/models/dpm/collisions/collision-partners/create

Creates a collision partner.

define/models/dpm/collisions/collision-partners/delete

Deletes a collision partner.

define/models/dpm/collisions/collision-partners/list

Lists all known collision partners.

define/models/dpm/collisions/collision-partners/rename

Renames a collision partner.

define/models/dpm/collisions/dem-collisions?

Enables/disables the DEM collision model.

define/models/dpm/collisions/list-all-pair-settings

For each pair of collision partners, lists the collision laws and their parameters.

define/models/dpm/collisions/max-particle-velocity

Sets the maximum particle velocity that may arise from collisions.

define/models/dpm/erosion-dynamic-mesh/

Enters the menu to enable/configure/run the erosion-dynamic mesh interaction.

define/models/dpm/erosion-dynamic-mesh/enable-erosion-dynamic-mesh-coupling?

Enables mesh deformation due to wall erosion.

define/models/dpm/erosion-dynamic-mesh/general-parameters/

Enters the menu for setting erosion coupling with dynamic mesh.

define/models/dpm/erosion-dynamic-mesh/general-parameters/dynamic-mesh-settings

Sets parameters for dynamic mesh calculations.

define/models/dpm/erosion-dynamic-mesh/general-parameters/erosion-settings

Sets parameters for erosion calculations.

define/models/dpm/erosion-dynamic-mesh/general-parameters/participating-walls

Specifies all participating walls.

define/models/dpm/erosion-dynamic-mesh/run-parameters/

Manages erosion-dynamic mesh run settings.

define/models/dpm/erosion-dynamic-mesh/run-parameters/autosave-files

Sets the iteration increment and filename to save data files.

define/models/dpm/erosion-dynamic-mesh/run-parameters/autosave-graphics

Sets the iteration increment to save graphics files.

define/models/dpm/erosion-dynamic-mesh/run-parameters/flow-simulation-control

Sets the number of iterations per flow simulation step.

define/models/dpm/erosion-dynamic-mesh/run-parameters/mesh-motion-time-step

Sets the mesh motion time stepping parameters and method.

define/models/dpm/erosion-dynamic-mesh/run-parameters/simulation-termination

Sets the total time of erosion.

define/models/dpm/erosion-dynamic-mesh/run-simulation

Performs a coupled erosion-dynamic mesh simulation.

define/models/dpm/injections/

Enters the injections menu.

define/models/dpm/injections/create-injection

Creates an injection.

define/models/dpm/injections/delete-injection

Deletes an injection.

define/models/dpm/injections/injection-properties/

Enters the menu to set or display properties for one or more injections.

define/models/dpm/injections/injection-properties/file/

Enters the menu to specify file injection settings.

define/models/dpm/injections/injection-properties/file/filename

Specifies the name for the injection file.

define/models/dpm/injections/injection-properties/file/repeat-interval-in-file

Puts the unsteady file injection into a time-periodic mode. For more information, see User Input for File Injections in the *Fluent User's Guide*.

define/models/dpm/injections/injection-properties/file/start-flow-time-in-file

Specifies the flow-time in the unsteady injection file from which Ansys Fluent starts reading the injection file. For more information, see User Input for File Injections in the *Fluent User's Guide*.

define/models/dpm/injections/injection-properties/list/

Enters the menu to display the current properties for one or more injections.

define/models/dpm/injections/injection-properties/list/list-picked-injections

Lists the injections that have been selected for display using `pick-injections-to-list`.

define/models/dpm/injections/injection-properties/list/list-picked-properties-to-list

Lists the properties that have been selected for display using `pick-properties-to-list`.

define/models/dpm/injections/injection-properties/list/list-property-values

Lists the settings for the properties selected with `pick-properties-to-list`. This text command is only available when a single injection has been selected with `pick-injections-to-list`. This option is available if at least two injections have been selected.

define/models/dpm/injections/injection-properties/list/list-uniform-values

Lists 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`. This option is available if at least two injections have been selected.

define/models/dpm/injections/injection-properties/list/list-values-per-injection

For each injection selected with `pick-injections-to-list`, lists the values for properties selected with `pick-properties-to-list`. This option is available if at least two injections ave been selected.

define/models/dpm/injections/injection-properties/list/list-values-per-property

For each property selected with `pick-properties-to-list`, lists the values for the injections selected with `pick-injections-to-list`. This option is available if at least two injections have been selected.

define/models/dpm/injections/injection-properties/list/pick-injections-to-list

Selects the injection or injections for which property values will be displayed. The use of asterisks (*) as wildcards is supported.

define/models/dpm/injections/injection-properties/list/pick-properties-to-list

Selects the property or properties for which values will be displayed. The use of asterisks (*) as wildcards is supported.

define/models/dpm/injections/injection-properties/set/

Enters the menu to set physical models such as drag and breakup for one or more injections.

define/models/dpm/injections/injection-properties/set/list-picked-injections

Lists the injections selected for setting with `pick-injections-to-set`.

define/models/dpm/injections/injection-properties/set/location/

Enters the injection location settings menu. This command is only available when you select the injection(s) using the prompts of the `define/models/dpm/injections/properties/set/pick-injections-to-set` text command.

define/models/dpm/injections/injection-properties/set/location/reference-frame

Sets the reference frame for injection point properties. You can choose either `global` (default) or one of the local reference frames that you created. This option is available only if at least one injection has been selected using `define/injections/properties/set/pick-injections-to-set`.

define/models/dpm/injections/injection-properties/set/location/spatial-staggering/

Enters the injection spatial-staggering settings menu.

define/models/dpm/injections/injection-properties/set/location/spatial-staggering/stagger-atomizer-positions?

Enables spatial staggering of the atomizer injection. This option is available only if at least one atomizer injection has been selected using `define/injections/properties/set/pick-injections-to-set`.

define/models/dpm/injections/injection-properties/set/location/spatial-staggering/stagger-radius

Specifies the stagger radius of the region from which particles are released. This option is available only if at least one standard injection has been selected using `define/injections/properties/set/pick-injections-to-set`.

define/models/dpm/injections/injection-properties/set/location/spatial-staggering/stagger-std-inj-positions?

Enables injection-specific spatial staggering of the particles. This option is available only if at least one standard injection has been selected using `define/injections/properties/set/pick-injections-to-set`.

define/models/dpm/injections/injection-properties/set/physical-models/

Enters the menu to set physical models such as drag and breakup for the selected injections. This command is only available when you select the injection(s) using the prompts of the `define/models/dpm/injections/properties/set/pick-injections-to-set` text command.

define/models/dpm/injections/injection-properties/set/physical-models/brownian-motion

Enables Brownian motion effects for the currently selected injection(s).

define/models/dpm/injections/injection-properties/set/physical-models/drag-parameters

Sets the drag law and corresponding parameters for the currently selected injection(s).

define/models/dpm/injections/injection-properties/set/physical-models/particle-rotation

Enters the menu to set-up rotation related models.

define/models/dpm/injections/injection-properties/set/physical-models/particle-rotation/enable-rotation

Enables/disables solution of ODE for the angular acceleration of particles.

define/models/dpm/injections/injection-properties/set/physical-models/particle-rotation/magnus-lift-law

Sets the law for the rotational lift coefficient used in the formulation of Magnus lift force.

define/models/dpm/injections/injection-properties/set/physical-models/particle-rotation/rotational-drag-law

Sets the law for the rotational drag coefficient.

define/models/dpm/injections/injection-properties/set/physical-models/rough-wall-model

Enables/disables the rough wall model.

define/models/dpm/injections/injection-properties/set/physical-models/spray-secondary-breakup

Enters 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.

define/models/dpm/injections/injection-properties/set/pick-injections-to-set

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

define/models/dpm/injections/list-particles

Lists particle streams in an injection.

define/models/dpm/injections/modify-all-injections/

Enters the menu to set properties for all injections.

define/models/dpm/injections/modify-all-injections/injection-type

Defines injection type.

define/models/dpm/injections/modify-all-injections/number-of-tries

Sets the number of stochastic tries.

define/models/dpm/injections/modify-all-injections/random-eddy-life-time?

Turns enable/disable a random eddy lifetime.

define/models/dpm/injections/modify-all-injections/stochastic-tracking?

Turns enable/disable stochastic tracking.

define/models/dpm/injections/modify-all-injections/time-scale-constant

Sets the time scale constant.

define/models/dpm/injections/rename-injection

Renames an injection.

define/models/dpm/injections/set-injection-properties

Sets injection properties.

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

define/models/dpm/interaction/

Sets parameters for coupled discrete phase calculations.

define/models/dpm/interaction/coupled-calculations?

Selects whether or not to couple continuous and discrete phase calculations.

define/models/dpm/interaction/dpm-iteration-interval

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

define/models/dpm/interaction/implicit-momentum-coupling?

Enables/disables implicit treatment for the DPM momentum source terms.

define/models/dpm/interaction/implicit-source-term-coupling?

Enables/disables implicit treatment for all DPM source terms.

`define/models/dpm/interaction/`linear-growth-of-dpm-source-term?

Enables/disables the linear ramping up of the DPM source terms at every DPM iteration.

`define/models/dpm/interaction/`linearized-dpm-mixture-fraction-source-terms?

Enables/disables linearization of mixture fraction source terms. This command is available only for non- or partially-premixed combustion cases.

`define/models/dpm/interaction/`linearized-dpm-source-terms?

Enables/disables linearization of source terms for the discrete phase.

`define/models/dpm/interaction/`replace-dpm-mass-source-by-mixture-fraction?

When enabled, recalculates the mixture fraction source terms as a function of the primary mixture fraction. This command is available for non- or partially-premixed combustion cases only.

`define/models/dpm/interaction/`reset-sources-at-timestep?

Enables/disables flush of DPM source terms at beginning of every time step.

`define/models/dpm/interaction/`underrelaxation-factor

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

`define/models/dpm/interaction/`update-dpm-sources-every-flow-iteration?

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

`define/models/dpm/`numerics/

Enters the numerics menu to set numerical solution parameters.

`define/models/dpm/numerics/`automated-scheme-selection?

Enables/disables the adaptation of integration step length based on a maximum error.

`define/models/dpm/numerics/`average-DDPM-variables?

Enables/disables mesh node averaging of DDPM quantities.

`define/models/dpm/numerics/`average-each-step?

Enables/disables mesh node averaging during integration time step.

`define/models/dpm/numerics/`average-kernel

Specifies the averaging kernel to use for mesh node averaging.

`define/models/dpm/numerics/`average-source-terms?

Enables/disables mesh node averaging of DPM source terms.

define/models/dpm/numerics/coupled-heat-mass-update

Enables/disables coupled heat and mass update.

define/models/dpm/numerics/drag-law

Sets the drag law.

define/models/dpm/numerics/enable-node-based-averaging?

Enables/disables mesh node averaging of DPM quantities.

define/models/dpm/numerics/error-control?

Adapts integration step length based on a maximum error.

define/models/dpm/numerics/gaussian-factor

Specifies the Gaussian constant when using the**gaussian** kernel for mesh node averaging.

define/models/dpm/numerics/high-resolution-tracking/

Enters the high resolution tracking menu. See Tracking Settings for the Discrete Phase Model in the *Fluent User's Guide* for more information about these options.

define/models/dpm/numerics/high-resolution-tracking/always-use-face-centroid-with-periodics?

When enabled, Ansys Fluent uses quad face centroids when creating subtets in cases with periodic boundaries.

define/models/dpm/numerics/high-resolution-tracking/barycentric-interpolation/

Enter the barycentric interpolation menu.

define/models/dpm/numerics/high-resolution-tracking/barycentric-interpolation/interpolate-flow-cp?

Enables/disables the barycentric interpolation of specific heat to the particle position. This option is recommended when the specific heat varies with position to avoid discontinuities in the interpolated variable at cell boundaries. For flows with constant specific heat, this option is unnecessary.

define/models/dpm/numerics/high-resolution-tracking/barycentric-interpolation/interpolate-flow-density?

Enables/disables the barycentric interpolation of the flow density. This option is recommended when the density varies with position to avoid discontinuities in the interpolated variable at cell boundaries. For constant density flows, this option is unnecessary.

define/models/dpm/numerics/high-resolution-tracking/barycentric-interpolation/interpolate-flow-solution-gradients?

When enabled, flow solution gradients are interpolated to the particle position. This can be useful when using physical models that depend on these gradients (for example, the thermophoretic force, pressure-gradient force, or virtual mass force). Interpolating the gradients also improves the accuracy and robustness of the trapezoidal numerics scheme, which is the default method for pathlines.

define/models/dpm/numerics/high-resolution-tracking/barycentric-interpolation/interpolate-flow-viscosity?

Enables/disables the barycentric interpolation of flow viscosity to the particle position. This option is recommended when the flow viscosity varies with position to avoid discontinuities in the interpolated variable at cell boundaries. For flows with constant viscosity, this option is unnecessary.

define/models/dpm/numerics/high-resolution-tracking/barycentric-interpolation/interpolate-temperature?

Enables/disables the barycentric interpolation of temperature to the particle position. The cell temperature is used by default in calculations of heat transfer to/from the particle.

define/models/dpm/numerics/high-resolution-tracking/barycentric-interpolation/interpolate-wallfilm-properties?

When enabled, the wall film properties (film height, film mass, and wall shear) are interpolated to the particle position.

define/models/dpm/numerics/high-resolution-tracking/barycentric-interpolation/zero-nodal-velocity-on-walls?

When enabled, sets the velocity at wall nodes to zero. (By default, the nodal velocities on walls are first reconstructed from cell and face values and then corrected to ensure that there are no velocity components directed towards the walls). This may be useful if you want to consider particle impingement on the walls. Note that enabling this option will more likely produce incomplete particles as some particles may settle on the walls.

define/models/dpm/numerics/high-resolution-tracking/barycentric-interpolation/use-legacy-particle-location-method?

Provides the ability to use the legacy particle location method with high-resolution tracking to improve performance. This option allows the faster but less accurate particle location method to be used with high-resolution tracking. This option is useful when tracking in polyhedral meshes, particularly when unsteady particles that evaporate quickly are injected frequently (for example, gas turbine combustors). Note that it does not improve the particle tracking speed, only the time it takes to inject the particles.

`define/models/dpm/numerics/high-resolution-tracking/`boundary-layer-tracking?

Enables/disables the calculation of the particle time step that considers both the cell aspect ratio and the particle trajectory. This method improves the accuracy of the predictions in boundary layer cells, particularly in layers where flow gradients are large.

`define/models/dpm/numerics/high-resolution-tracking/`check-subtet-validity?

When enabled, checks the validity of a subtet when the particle first enters it. If the subtet is found to be degenerate, the tracking algorithm modifies to accommodate it.

`define/models/dpm/numerics/high-resolution-tracking/`enable-automatic-intersection-tolerance?

Enables/disables the automatic calculation of intersection tolerance. By default, the tolerance used in intersection calculations is scaled by the residence time of the particle in the cell to improve robustness. For most cases, the scaled tolerance is sufficient to identify all intersections of the particle trajectory and the subtet faces. You can set the intersection tolerance manually using the `set-subtet-inter-section-tolerance` text command.

`define/models/dpm/numerics/high-resolution-tracking/`enable-barycentric-intersections?

Enables/disables an alternative method of calculating intersections with cell boundaries. Barycentric intersections are linear calculations and are faster than the default intersection algorithm. The default intersection algorithm is second-order for stationary meshes; therefore, using the barycentric intersection may sacrifice accuracy. You must verify that the barycentric intersections provide comparable results to the default intersection method. This option is available only for 3D stationary meshes and the double precision solver.

`define/models/dpm/numerics/high-resolution-tracking/`enable-high-resolution-tracking?

Enables/disables high resolution tracking.

`define/models/dpm/numerics/high-resolution-tracking/`enhanced-wallfilm-location-method?

When enabled, Ansys Fluent uses a more robust method of locating the wall film particles on the film zone after the domain has been remeshed either due to dynamic mesh movement or automatic adaption. The improved robustness involves a performance penalty. This method is recommended for cases when Fluent reports that wall film particles are lost during relocation.

define/models/dpm/numerics/high-resolution-tracking/project-wall-film-particles-to-film?

Enables/disables projecting existing particles to Lagrangian wall film to track using high-resolution tracking.

When reading in a data file that contains wall film particles previously tracked with the existing Ansys Fluent tracking method, you need to either clear the particles from the domain or project their positions to the wall film surface using the `project-wall-film-particles-to-film?` text command prior to using the high-resolution tracking method. After tracking the particles for one timestep, this option can be disabled to improve performance.

define/models/dpm/numerics/high-resolution-tracking/ set-film-spreading-parameter

Provides the ability to add a random component to the Lagrangian wall-film particle acceleration to reduce streaking that may result from tracking particles on faceted geometry. As small variations in face normals can produce streaks in the wall-film appearance, this parameter makes the film look smoother. The value of this parameter should range between 0 and 1.0. The default value of 0 prevents any artificial smoothing.

define/models/dpm/numerics/high-resolution-tracking/set-subtet-intersection-tolerance

Specifies the tolerance used in intersection calculations. This tolerance will be scaled by the characteristic cell crossing time of the particle if the `enable-automatic-intersection-tolerance?` text command is enabled. If that option is disabled, the specified tolerance will be used without scaling. The default intersection tolerance is 10^{-5}.

define/models/dpm/numerics/high-resolution-tracking/sliding-interface-crossover-fraction

Specifies the fraction of the distance to the subtet center to move the particle.

At non-conformal interfaces, the nodes used for the barycentric interpolation are different on either side of the interface. This may result in incomplete particles due to discontinuities in the variable interpolation. The number of incomplete particles may be reduced by moving the particles slightly off of the sliding interface. Recommended values range between 0 and 0.5.

define/models/dpm/numerics/high-resolution-tracking/use-barycentric-sampling?

When enabled, this option provides improved accuracy and parallel consistency when sampling particles at planes. This item is available only with the 3D solver. Using the double-precision solver and bounded planes is recommended.

define/models/dpm/numerics/high-resolution-tracking/use-quad-face-centroid?

Enables/disables using quad face centroids when creating subtets. This option changes the way hexahedral cells are decomposed to avoid creating degenerate subtets.

define/models/dpm/numerics/high-resolution-tracking/use-particle-timestep-for-intersection-tolerance?

Enables/disables the use of the particle timestep for the subtet intersection tolerance with axisymmetric grids (default: enabled). If disabled, the tolerance will be calculated in the same manner as non-axisymmetric meshes (a scaled value of the tolerance which is set using the `define/models/dpm/numerics/high-resolution-tracking/set-subtet-intersection-tolerance` text command).

define/models/dpm/numerics/high-resolution-tracking/use-velocity-based-error-control?

Enables/disables an alternative method of timestep adaption. By default, Ansys Fluent uses the half-step method of timestep adaption with particle integration. This alternative method of controlling the integration timestep based upon velocity changes is faster; however, you need to ensure that the accuracy is comparable for your specific application.

define/models/dpm/numerics/minimum-liquid-fraction

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

define/models/dpm/numerics/re-randomize-every-iteration?

Determines whether or not the initial particle positions will vary from iteration to iteration.

define/models/dpm/numerics/re-randomize-every-timestep?

Determines whether or not the initial particle positions will vary from time step to time step.

define/models/dpm/numerics/tracking-parameters

Sets parameters for the (initial) tracking step length.

define/models/dpm/numerics/tracking-scheme

Specifies a tracking scheme.

define/models/dpm/numerics/tracking-statistics

Controls the format of the one-line tracking statistics to be printed after every DPM tracking pass. A value of **0** (the default) prints only fates with non-zero values. A value of **1** prints all fates, including fates with zero values.

define/models/dpm/numerics/underrelax-film-height

Sets the under-relaxation factor for the film height calculation. The recommended values range between 0.5 (default) and 0.9.

define/models/dpm/numerics/vaporization-limiting-factors

Sets the Vaporization Fractional Change Limits.

define/models/dpm/options/

Enters the options menu to set optional models.

define/models/dpm/options/allow-supercritical-pressure-vaporization?

Enforces the switching from vaporization to boiling even if the boiling point is not calculated from the vapor pressure data. If the pressure in your model is above critical you must retain the default setting (`yes`). This options is available only if when**Pressure Dependent Boiling** is enabled in the **Physical Models** tab of the **Discrete Phase Models** dialog box. For more details, see Enabling Pressure Dependent Boiling in the *Fluent User's Guide*.

define/models/dpm/options/brownian-motion

Enables/disables Brownian motion of particles.

define/models/dpm/options/current-positions-in-sample-file-format?

When enabled, generates a file containing particle current positions (step-by-step history report for unsteady tracking) in the sampling file format.

define/models/dpm/options/enable-contour-plots

Enables computation of mean and/or RMS values of additional discrete phase variables for postprocessing.

define/models/dpm/options/ensemble-average

Ensembles average cloud properties.

define/models/dpm/options/erosion-accretion

Enables/disables erosion/accretion.

define/models/dpm/options/init-erosion-accretion-rate

Initializes the erosion/accretion rates with zero.

define/models/dpm/options/maximum-udf-species

Specifies 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.

define/models/dpm/options/particle-radiation

Enables/disables particle radiation.

define/models/dpm/options/pressure-gradient-force

Enables/disables inclusion of pressure gradient effects in the particle force balance.

define/models/dpm/options/remove-wall-film-temperature-limiter?

Answering **yes** at the prompt removes the wall temperature limiter for Lagrangian wall-film walls. If you enter **no** (default), two additional prompts will appear in the console allowing you to define the temperature difference above the boiling point and to enable/disable the reporting of the Leidenfrost temperature on the wall faces.

define/models/dpm/options/scr-urea-deposition-risk-analysis/

Enters the menu for setting up the risk for solids deposit formation for the Selective Catalytic Reduction (SCR) process. For more information, see Assessing the Risk for Solids Deposit Formation During Selective Catalytic Reduction Process in the *Fluent User's Guide*.

define/models/dpm/options/scr-urea-deposition-risk-analysis/cryst-depo-weight

Sets the weighting factor for crystallization reactions risk.

define/models/dpm/options/scr-urea-deposition-risk-analysis/cryst-max-temp

Sets maximum temperature for urea crystallization.

define/models/dpm/options/scr-urea-deposition-risk-analysis/cryst-min-mass-fract

Sets the minimum urea mass fraction for crystallization.

define/models/dpm/options/scr-urea-deposition-risk-analysis/cryst-min-temp

Sets minimum temperature for urea crystallization.

define/models/dpm/options/scr-urea-deposition-risk-analysis/enable?

Enables/disables the SCR urea deposition risk analysis.

define/models/dpm/options/scr-urea-deposition-risk-analysis/heat-flux-based-risk-weight

Sets the weighting factor for heat flux-based component within hydrodynamic risk.

define/models/dpm/options/scr-urea-deposition-risk-analysis/hydro-dynamic-risk-weight

Sets the weighting factor for all hydrodynamic deposition risk.

define/models/dpm/options/scr-urea-deposition-risk-analysis/seco-rx-max-temp

Sets maximum temperature for secondary reactions.

define/models/dpm/options/scr-urea-deposition-risk-analysis/seco-rx-min-hnco

Sets the minimum HNCO mass fraction in the gas phase above the film for secondary reactions.

define/models/dpm/options/scr-urea-deposition-risk-analysis/seco-rx-min-temp

Sets minimum temperature for secondary reactions.

define/models/dpm/options/scr-urea-deposition-risk-analysis/velocity-based-risk-weight

Sets the weighting factor for velocity-based component within hydrodynamic risk.

define/models/dpm/options/scr-urea-deposition-risk-analysis/wall-face-zones

Lists the selected wall zones and allows you to modify the selection list.

define/models/dpm/options/set-thermolysis-limit

Sets the limit for the thermolysis model.

define/models/dpm/options/stagger-spatially-atomizer-injections?

Enables/disables spatial staggering for atomizer and solid-cone injections.

define/models/dpm/options/stagger-spatially-standard-injections?

Enables/disables spatial staggering for standard (non-atomizer and non-solid-cone) injections.

define/models/dpm/options/stagger-temporally?

Enables/disables temporal staggering.

define/models/dpm/options/staggering-factor

Controls the percentage of every particle's initial time step that will be sampled.

define/models/dpm/options/saffman-lift-force

Enables/disables Saffman lift force.

define/models/dpm/options/stagger-radius

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

define/models/dpm/options/step-report-sig-figures

Sets significant figures in the step-by-step report.

define/models/dpm/options/thermophoretic-force

Enables/disables thermophoretic force.

define/models/dpm/options/track-in-absolute-frame

Enables/disables tracking in absolute frame.

define/models/dpm/options/two-step-devolatilization-limit

Sets the limit for the two-step devolatilization reaction.

define/models/dpm/options/treat-multicomponent-saturation-temperature-failure?

Enables/disables dumping multicomponent particle mass into the continuous phase if the saturation temperature calculation fails.

define/models/dpm/options/two-way-coupling

Enables/disables calculation of DPM sources in TKE equation.

define/models/dpm/options/uniform-mass-distribution-for-injections?

Specifies 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.

define/models/dpm/options/use-absolute-pressure-for-vaporization?

Determines whether the absolute pressure or constant operating pressure (specified in `define/operating-conditions/operating-pressure`) will be used in vaporization rates calculations.

define/models/dpm/options/vaporization-heat-transfer-averaging

Enables averaging of the Spalding heat transfer term for the convection/diffusion-controlled model.

define/models/dpm/options/vaporization-options

Sets Vaporization options.

define/models/dpm/options/virtual-mass-force

Enables/disables inclusion of the virtual mass force in the particle force balance.

define/models/dpm/parallel/

Enters the parallel menu to set parameters for parallel DPM calculations.

define/models/dpm/parallel/enable-workpile?

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

define/models/dpm/parallel/expert/

Enters the menu for expert DPM parallel text commands.

define/models/dpm/parallel/expert/partition-method-hybrid-2domain

Enables/disables a partitioning method that is more granular and can yield faster calculations (especially for cases that are running on a low to moderate number of processors). This partitioning method is only applied when you use the DPM domain for the hybrid parallel DPM tracking mode (that is, when you have enabled the `define/models/dpm/parallel/hybrid-2domain?` text command).

define/models/dpm/parallel/hybrid-2domain?

Enables/disables the use of a second domain for DPM particle tracking.

define/models/dpm/parallel/n-threads

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

define/models/dpm/parallel/report

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

define/models/dpm/parallel/use-hybrid

Specifies 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.

define/models/dpm/parallel/use-message-passing

Specifies 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.

define/models/dpm/parallel/use-shared-memory

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

define/models/dpm/splash-options/

Enters the splash option menu.

define/models/dpm/splash-options/orourke-splash-fraction

Enables/disables the O'Rourke formulation (default for the Lagrangian Wall Film (LWF) model). If the O'Rourke formulation is disabled, the Stanton formulation (default for the Eulerian Wall Film (EWF) model) is used in a simulation.

define/models/dpm/splash-options/splash-pdf-limiting

Sets the splash pdf limiting method. Available methods are: the splash pdf tail limiting (default for the LWF model) and the splash pdf peak limiting (default for the EWF model). For the splash pdf peak limiting, you will be prompted to specify the peak limiting value.

define/models/dpm/spray-model/

Enters the spray model menu. This command is available only if the breakup model enabled globally.

define/models/dpm/spray-model/breakup-model-summary

Current spray model settings.

define/models/dpm/spray-model/consider-children-in-the-same-tracking-step?

Enables/disables collecting and tracking new generated child droplets within the same time step.

define/models/dpm/spray-model/droplet-coalescence?

Enables/disables droplet coalescence when using the stochastic collision model. This option is available if all injections have been selected and the DEM model is disabled.

define/models/dpm/spray-model/droplet-collision?

Enables/disables droplet collision model.

define/models/dpm/spray-model/enable-breakup?

Enables/disables breakup globally, but does not alter injection settings other than enable/disable.

define/models/dpm/spray-model/khrt-model

Sets the KHRT breakup model.

define/models/dpm/spray-model/madabhushi-model

Sets the Madabhushi breakup model.

define/models/dpm/spray-model/no-breakup

Disables the currently enabled breakup model. This option is available only if the breakup model is enabled globally and for the selected injections, and not all injections have been selected.

define/models/dpm/spray-model/schmehl-model

Sets the Schmehl breakup model.

define/models/dpm/spray-model/set-breakup

Enables/disables breakup model globally and uniformly specifies injection breakup parameters.

define/models/dpm/spray-model/ssd-model

Sets the SSD breakup model.

define/models/dpm/spray-model/tab-model

Sets the TAB breakup model.

define/models/dpm/spray-model/tab-number-of-breakup-parcels

Sets the number of parcels to break up a droplet in the TAB model.

define/models/dpm/spray-model/tab-randomize-breakup-parcel-diameter?

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

define/models/dpm/spray-model/wave-allow-rayleigh-growth?

Allows treatment of the Rayleigh regime, in which a cylindrical liquid jet breaks into droplets of larger diameter. This option is available only if the WAVE model is enabled.

define/models/dpm/spray-model/wave-mass-cutoff

Sets the minimum percentage of parent parcel mass shed before new parcel creation. This option is available only if the WAVE model is enabled.

define/models/dpm/spray-model/wave-model

Sets the WAVE breakup model.

define/models/dpm/spray-model/wave-spray-angle-constant

Sets the spray-angle constant to compute orthogonal velocity components of child droplets after breakup. This option is available only if the WAVE model is enabled.

define/models/dpm/stripping-options/

Enters the stripping options menu.

define/models/dpm/stripping-options/diameter-coefficient

Sets the diameter coefficient (F in Equation 18.20 in the Theory Guide)

`define/models/dpm/stripping-options/mass-coefficient`

Sets the mass coefficient (C in Equation 18.21 in the Theory Guide)

`define/models/dpm/unsteady-tracking`

Enables/disables unsteady particle tracking.

`define/models/dpm/user-defined`

Sets DPM user-defined functions.

`define/models/potential-and-li-ion-battery?`

Enables/disables the electric-potential model.

`define/models/energy?`

Enables/disables the energy model.

`define/models/eulerian-wallfilm/`

Enters the Eulerian wall film model menu.

`define/models/eulerian-wallfilm/coupled-solution/`

Enters the Coupled-Solution menu.

`define/models/eulerian-wallfilm/coupled-solution/enable-coupled-solution?`

Enables/disables the coupled solution method.

`define/models/eulerian-wallfilm/coupled-solution/enable-curvature-smoothing?`

Enables/disables the film curvature smoothing option and sets the smoothing parameters.

`define/models/eulerian-wallfilm/enable-wallfilm-model?`

Enables/disables Eulerian Wall Film Model.

`define/models/eulerian-wallfilm/initialize-wallfilm-model`

Initializes Eulerian Wall Film Model.

`define/models/eulerian-wallfilm/solve-wallfilm-equation?`

Activates Eulerian Wall Film Equations.

`define/models/eulerian-wallfilm/model-options`

Sets Eulerian Wall Film Model Options.

`define/models/eulerian-wallfilm/film-material`

Sets Film Material and Properties.

`define/models/eulerian-wallfilm/`solution-options

Sets Eulerian Wall Film Model Solution Options.

`define/models/`frozen-flux?

Enables/disables frozen flux formulation for transient flows.

`define/models/`heat-exchanger/

Enters the heat exchanger menu.

`define/models/heat-exchanger/`dual-cell-model/

Enters the dual cell model menu.

`define/models/heat-exchanger/dual-cell-model/`add-heat-exchanger

Adds heat-exchanger.

`define/models/heat-exchanger/dual-cell-model/`alternative-formulation?

Enables/disables alternative formulation for heat transfer calculations.

`define/models/heat-exchanger/dual-cell-model/`delete-heat-exchanger

Deletes heat-exchanger.

`define/models/heat-exchanger/dual-cell-model/`heat-exchanger?

Enables/disables the dual cell heat-exchanger model.

`define/models/heat-exchanger/dual-cell-model/`modify-heat-exchanger

Modifies heat-exchanger.

`define/models/heat-exchanger/dual-cell-model/`plot-NTU

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

`define/models/heat-exchanger/dual-cell-model/`write-NTU

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

`define/models/heat-exchanger/`macro-model/

Enters the heat macro-model menu.

`define/models/heat-exchanger/macro-model/`delete-heat-exchanger-group

Deletes heat-exchanger group.

`define/models/heat-exchanger/macro-model/`heat-exchanger?

Enables/disables heat-exchanger model.

define/models/heat-exchanger/macro-model/heat-exchanger-group

Defines heat-exchanger group.

define/models/heat-exchanger/macro-model/heat-exchanger-macro-report

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

define/models/heat-exchanger/macro-model/heat-exchanger-model

Defines heat-exchanger core model.

define/models/heat-exchanger/macro-model/heat-exchanger-report

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

define/models/heat-exchanger/macro-model/heat-exchanger-zone

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

define/models/heat-exchanger/macro-model/plot-NTU

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

define/models/heat-exchanger/macro-model/write-NTU

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

define/models/multiphase/

Enters the multiphase model menu.

define/models/multiphase/body-force-formulation

Specifies body force formulation.

define/models/multiphase/boiling-model-options

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

define/models/multiphase/coupled-level-set

Enables coupled level set interface tracking method.

define/models/multiphase/eulerian-parameters

Specifies Eulerian parameters.

define/models/multiphase/explicit-expert-options/

Enters the menu to set explicit VOF expert options.

define/models/multiphase/explicit-expert-options/solve-vof-every-iter?

If you enter **yes**, the volume fraction equations will be solved every iteration. By default, the volume fraction equations will be solved only once per time step.

define/models/multiphase/explicit-expert-options/sub-time-step-method

Selects the sub-time step method.

define/models/multiphase/explicit-expert-options/volume-fraction-filtering/

Enters the volume fraction filtering menu.

define/models/multiphase/explicit-expert-options/volume-fraction-filtering/enable?

Enables/disables the volume fraction filtering treatment.

define/models/multiphase/explicit-expert-options/volume-fraction-filtering/filtering-options

Selects the volume fraction filtering method. This command becomes available once the define/models/multiphase/explicit-expert-options/volume-fraction-filtering/enable? text option has been set to yes.

define/models/multiphase/explicit-expert-options/volume-fraction-filtering/vol-frac-cutoff

Specifies a cut-off value for the volume fraction filtering. This command becomes available after you select the node averaged cutoff method using the define/models/multiphase/explicit-expert-options/volume-fraction-filtering/filtering-options text command.

define/models/multiphase/interface-modeling-options

Specifies interface modeling options.

define/models/multiphase/mixture-parameters

Specifies mixture parameters.

define/models/multiphase/model

Specifies multiphase model.

define/models/multiphase/number-of-phases

Specifies the number of phases.

define/models/multiphase/population-balance/

Enters the population balance models menu.

define/models/multiphase/population-balance/includeexpansion?

Allows you to account for bubble expansion due to large changes in hydrostatic pressure in compressible flows. This option is available with the inhomogeneous discrete method.

define/models/multiphase/population-balance/model

Allows you to select the population balance model and set its parameters.

define/models/population-balance/phenomena/

Enters the phenomena menu for population balance.

define/models/population-balance/phenomena/aggregation

Sets the aggregation kernel.

define/models/multiphase/population-balance/phenomena/aggregation-factor

Specifies a factor that controls the intensity of the selected aggregation kernel.

define/models/multiphase/population-balance/phenomena/breakage

Sets the breakage kernel.

define/models/multiphase/population-balance/phenomena/breakage-factor

Specifies a factor that controls the intensity of the selected breakage kernel.

define/models/multiphase/population-balance/phenomena/growth

Specifies the growth rate.

define/models/multiphase/population-balance/phenomena/nucleation

Specifies the nucleation rate.

define/models/multiphase/population-balance/size-calculator

Gives you recommendations for appropriate bubble sizes and/or droplet size limits.

define/models/multiphase/regime-transition-modeling

enables the Algebraic Interfacial Area Density (AIAD) model and sets the AIAD secondary continuous phase and the secondary entrained phase. Entering `0` as a phase ID cancels any previous phase selection. Note that you must define the phases in your simulation using the `define/phases/` text command prior to using the `regime-transition-modeling` text command. This option is available only with the Eulerian multiphase model.

define/models/multiphase/vof-sub-models

Enables the Open Channel sub-model and/or the Open Channel Wave Boundary Condition sub-model.

define/models/multiphase/volume-fraction-parameters

Specifies volume fraction parameters.

define/models/multiphase/wet-steam/

Enters the wet steam model menu.

define/models/multiphase/wet-steam/compile-user-defined-wetsteam-functions

Compiles user-defined wet steam library.

define/models/multiphase/wet-steam/enable?

Enables/disables the wet steam model.

define/models/multiphase/wet-steam/load-unload-user-defined-wetsteam-library

Loads or unloads user-defined wet steam library.

define/models/multiphase/wet-steam/set/

Enters the set menu for setting wet steam model options.

define/models/multiphase/wet-steam/set/max-liquid-mass-fraction

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

define/models/multiphase/wet-steam/set/droplet-growth-rate

Sets the droplet growth rate to either Young's formulation (default) or Hill's.

define/models/multiphase/wet-steam/set/virial-equation

Sets the equation of state for steam to either Vukalovich formulation (default) or Young formulation.

define/models/noniterative-time-advance?

Enables/disables noniterative time advancement scheme.

define/models/nox?

Enables/disables the NOx model.

define/models/nox-parameters/

Enters the NOx parameters menu.

define/models/nox-parameters/inlet-diffusion?

Enables/disables inclusion of diffusion at inlets.

define/models/nox-parameters/nox-chemistry

Selects NOx chemistry model.

define/models/nox-parameters/nox-expert

Selects additional NOx equations.

define/models/nox-parameters/nox-turbulence-interaction

Sets NOx turbulence interaction model.

define/models/radiation/

Enters the radiation models menu.

define/models/radiation/apply-full-solar-irradiation?

Enables/disables the application of the complete solar load to the first wavelength band only, reverting to the pre-2019 R1 behavior of the Solar Load and Discrete Ordinates models.

define/models/radiation/blending-factor

Sets 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.

define/models/radiation/discrete-ordinates?

Enables/disables discrete ordinates radiation model.

define/models/radiation/discrete-transfer?

Enables/disables discrete transfer radiation model.

define/models/radiation/do-acceleration?

Enables/disables the acceleration of the discrete ordinates (DO) radiation model calculations. Note that this text command is only available when running on Linux in parallel.

define/models/radiation/do-coupling?

Enables/disables DO/energy coupling.

define/models/radiation/solar-irradiation?

Enables/disables the solar irradiation model.

define/models/radiation/dtrm-parameters/

Enters the dtrm parameters menu.

define/models/radiation/dtrm-parameters/check-ray-file

Reads DTRM rays file.

define/models/radiation/dtrm-parameters/controls

Sets dtrm solution controls.

define/models/radiation/dtrm-parameters/make-globs

Makes globs (coarser mesh) for radiation.

define/models/radiation/dtrm-parameters/ray-trace

Creates DTRM rays for radiation.

define/models/radiation/fast-second-order-discrete-ordinate?

Enables/disables the fast-second-order option for Discrete Ordinate Model.

define/models/radiation/mc-model-parameters

Specifies Monte Carlo model parameters. This text command is available only when the Monte Carlo model is enabled.

define/models/radiation/mc-under-relaxation

Sets the under-relaxation factor for Monte Carlo radiation sources used in the energy equation.

define/models/radiation/method-partially-specular-wall

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

define/models/radiation/montecarlo?

Enables/disables the Monte Carlo radiation model.

define/models/radiation/non-gray-model-parameters

Sets parameters for non-gray model.

define/models/radiation/p1?

Enables/disables P1 radiation model.

define/models/radiation/radiation-iteration-parameters

Sets iteration parameters for radiation models.

define/models/radiation/radiation-model-parameters

Sets parameters for radiation models.

define/models/radiation/rosseland?

Enables/disables Rosseland radiation model.

define/models/radiation/s2s?

Enables/disables S2S radiation model.

define/models/radiation/s2s-parameters/

Enters the S2S parameters menu.

define/models/radiation/s2s-parameters/**compute-fpsc-values**

Computes only fpsc values based on current settings

define/models/radiation/s2s-parameters/**compute-vf-only**

Computes/writes view factors only.

define/models/radiation/s2s-parameters/**compute-write-vf**

Computes/writes surface clusters and view factors for S2S radiation model.

define/models/radiation/s2s-parameters/**enable-mesh-interface-clustering?**

enables surface clusters on mesh interfaces.

define/models/radiation/s2s-parameters/**non-participating-boundary-zones-temperature**

Sets temperature for the non-participating boundary zones.

define/models/radiation/s2s-parameters/**print-thread-clusters**

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

define/models/radiation/s2s-parameters/**print-zonewise-radiation**

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

define/models/radiation/s2s-parameters/**read-vf-file**

Reads S2S file.

define/models/radiation/s2s-parameters/**set-global-faces-per-surface-cluster**

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

define/models/radiation/s2s-parameters/**set-vf-parameters**

Sets the parameters needed for the viewfactor calculations.

define/models/radiation/s2s-parameters/**split-angle**

Sets split angle for the clustering algorithm.

define/models/radiation/s2s-parameters/use-new-cluster-algorithm

Uses the new surface clustering algorithm.

define/models/radiation/s2s-parameters/use-old-cluster-algorithm

Uses the old surface clustering algorithm.

define/models/radiation/solar?

Enables/disables solar model.

define/models/radiation/solar-calculator

Calculates sun direction and intensity.

define/models/radiation/solar-parameters/

Enters the solar parameters menu.

define/models/radiation/solar-parameters/autoread-solar-data

Sets autoread solar data parameters.

define/models/radiation/solar-parameters/autosave-solar-data

Sets autosave solar data parameters.

define/models/radiation/solar-parameters/ground-reflectivity

Sets ground reflectivity parameters.

define/models/radiation/solar-parameters/illumination-parameters

Sets illumination parameters.

define/models/radiation/solar-parameters/iteration-parameters

Sets update parameters.

define/models/radiation/solar-parameters/quad-tree-parameters

Sets quad-tree refinement parameters.

define/models/radiation/solar-parameters/scattering-fraction

Sets scattering fraction parameters.

define/models/radiation/solar-parameters/solar-thread-control

Sets the number of threads to run the solar flux calculation. This item appears only when running in parallel with nodes located on a separate machine from the one running the host process and **Solar Ray Tracing** is enabled.

define/models/radiation/solar-parameters/sol-adjacent-fluidcells

Sets solar load on for adjacent fluid cells.

define/models/radiation/solar-parameters/sol-camera-pos

Sets camera position based on sun direction vector.

define/models/radiation/solar-parameters/sol-on-demand

Sets solar load on demand.

define/models/radiation/solar-parameters/sun-direction-vector

Sets sun direction vector.

define/models/radiation/solar-parameters/use-direction-from-sol-calc

Sets direction computed from solar calculator.

define/models/radiation/solution-method-for-do-coupling

Enables/disables the solution method for DO/energy coupling.

define/models/radiation/target-cells-per-volume-cluster

Sets the amount of coarsening of the radiation mesh for the Monte Carlo radiation model. A number greater than one implies coarsening, whereas equal to one implies no coarsening.

define/models/radiation/wsggm-cell-based

Enables/disables 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.

define/models/shell-conduction/

Enters the shell conduction models menu.

define/models/shell-conduction/enhanced-encapsulation?

Enables/disables an enhanced routine for the encapsulation of coupled walls during mesh partitioning that is enabled by default when shell conduction and/or the surface to surface (S2S) radiation model is used.

define/models/shell-conduction/multi-layer-shell?

Enables/disables the ability to define multi-layer shell conduction for walls. Note that the warped-face gradient correction (WFGC) is not supported when multi-layer shells are disabled.

define/models/shell-conduction/read-csv

Defines the shell conduction settings by reading a CSV file.

define/models/shell-conduction/save-shell-zones?

Enables the saving of shell zones to case files.

define/models/shell-conduction/settings

Enables shell conduction and defines the settings for any wall or group of walls by manually entering the number and properties of the layers.

define/models/shell-conduction/write-csv

Writes your saved shell conduction settings to a CSV file.

define/models/solidification-melting?

Enables/disables the solidification and melting model.

define/models/solver/

Enters the menu to select the solver.

define/models/solver/density-based-explicit

Enables/disables the density-based-explicit solver.

define/models/solver/density-based-implicit

Enables/disables the density-based-implicit solver.

define/models/solver/pressure-based

Enables/disables the pressure-based solver.

define/models/soot?

Enables/disables the soot model.

define/models/soot-parameters/

Enters the soot parameters menu.

define/models/soot-parameters/inlet-diffusion?

Enables/disables inclusion of diffusion at inlets.

define/models/soot-parameters/modify-schmidt-number?

Changes the turbulent Schmidt number for soot/nuclei equations.

define/models/soot-parameters/soot-model-parameters

Selects soot model parameters.

define/models/soot-parameters/soot-process-parameters

Selects soot process parameters.

define/models/soot-parameters/soot-radiation-interaction

Enables/disables the soot-radiation interaction model.

define/models/soot-parameters/soot-turbulence-interaction

Sets soot-turbulence interaction model.

define/models/**species/**

Enters the species models menu.

define/models/species/**CHEMKIN-CFD?**

Enables/disables the Ansys CHEMKIN-CFD solver.

define/models/species/**CHEMKIN-CFD-parameters/**

Enters the expert CHEMKIN-CFD parameters menu.

define/models/species/CHEMKIN-CFD-parameters/**add-cell-monitor**

Monitors cell for debug output.

define/models/species/CHEMKIN-CFD-parameters/**advanced-options**

Sets advanced parameter options.

define/models/species/CHEMKIN-CFD-parameters/**basic-options**

Sets basic parameter options.

define/models/species/CHEMKIN-CFD-parameters/**delete-cell-monitors**

Deletes cell monitors.

define/models/species/CHEMKIN-CFD-parameters/**list-cell-monitors**

Lists cell monitors.

define/models/species/**clear-isat-table**

Clears ISAT table.

define/models/species/**coal-calculator**

Sets up coal modeling inputs.

define/models/species/**combustion-numerics**

Applies optimal solver settings automatically to provide a faster solution time. This command is available only for transient non-premixed and partially premixed combustion models.

define/models/species/**decoupled-detailed-chemistry?**

Enables/disables the Decoupled Detailed Chemistry model.

define/models/species/diffusion-energy-source?

Enables/disables diffusion energy source.

define/models/species/electro-chemical-surface-reactions?

Enables/disables electrochemical surface reactions.

define/models/species/epdf-energy?

Enables/disables EPDF energy option.

define/models/species/flamelet-expert

Sets flamelet expert parameters.

define/models/species/full-tabulation?

Enables/disables building of a full 2-mixture fraction table

define/models/species/heat-of-surface-reactions?

Enables/disables heat of surface reactions.

define/models/species/ignition-model?

Enables/disables the ignition model.

define/models/species/ignition-model-controls

Sets ignition model parameters.

define/models/species/import-flamelet-for-restart

Imports Flamelet File for Restart.

define/models/species/inert-transport-controls

Sets inert transport model parameters.

define/models/species/inert-transport-model?

Enables/disables the inert transport model.

define/models/species/inlet-diffusion?

Enables/disables inclusion of diffusion at inlets.

define/models/species/integration-parameters

Sets chemistry ODE integrator parameters. Enables/disables stiff chemistry acceleration methods and set their parameters.

define/models/species/init-unsteady-flamelet-prob

Initializes Unsteady Flamelet Probability.

define/models/species/liquid-energy-diffusion?

Enables/disables energy diffusion for liquid regime.

define/models/species/liquid-micro-mixing?

Enables/disables liquid micro mixing.

define/models/species/mass-deposition-source?

Enables/disables mass deposition source due to surface reactions.

define/models/species/mixing-model

Sets PDF Transport mixing model.

define/models/species/multicomponent-diffusion?

Enables/disables multicomponent diffusion.

define/models/species/non-premixed-combustion?

Enables/disables non-premixed combustion model.

define/models/species/non-premixed-combustion-expert

Sets PDF expert parameters.

define/models/species/non-premixed-combustion-parameters

Sets PDF parameters.

define/models/species/off?

Enables/disables solution of species models.

define/models/species/partially-premixed-combustion?

Enables/disables partially premixed combustion model.

define/models/species/partially-premixed-combustion-expert

Sets PDF expert parameters.

define/models/species/partially-premixed-combustion-grids

Sets values for the grid distribution for PDF table or flamelet parameter. This text command is available only for partially premixed combustion cases with FGM.

define/models/species/partially-premixed-combustion-parameters

Sets PDF parameters.

define/models/species/partially-premixed-properties

Sets/changes partially-premixed mixture properties. This command is only available when `partially-premixed-combustion?` is enabled.

define/models/species/re-cacl-par-premix-props

Re-calculates partially-premixed properties. This command is only available when `partially-premixed-combustion?` is enabled.

define/models/species/particle-surface-reactions?

Enables/disables particle surface reactions.

define/models/species/pdf-transport?

Enables/disables the composition PDF transport combustion model.

define/models/species/pdf-transport-expert?

Enables/disables PDF Transport expert user.

define/models/species/premixed-model

Sets premixed combustion model.

define/models/species/premixed-combustion?

Enables/disables premixed combustion model.

define/models/species/reaction-diffusion-balance?

Enables/disables reaction diffusion balance at reacting surface for surface reactions.

define/models/species/reacting-channel-model?

Enables/disables the Reacting Channel Model.

define/models/species/reacting-channel-model-options

Sets Reacting Channel Model parameters.

define/models/species/reactor-network-model?

Enables/disables the Reactor Network Model.

define/models/species/relax-to-equil?

Enables/disables the Relaxation to Chemical Equilibrium model.

define/models/species/save-gradients?

Enables/disables storage of species mass fraction gradients.

define/models/species/set-premixed-combustion

Sets premixed combustion parameters.

define/models/species/set-turb-chem-interaction

Sets EDC model constants.

define/models/species/spark-model

Switches between the R15 and R14.5 spark models and sets spark model parameters.

define/models/species/species-migration?

Includes species migration in electric field. This command is available only when the electro-chemical surface reactions are enabled.

define/models/species/species-transport-expert/

Sets the convergence acceleration expert parameters. This command is only available when the species transport model is enabled.

define/models/species/species-transport-expert/useconvergence acceleration method?

Answeringyes at the prompt, uses a combination of different convergence acceleration techniques. If you enterno (default), additional options will be displayed in the console.

define/models/species/species-transport-expert/Usespecies clipping?

Enables/disables the explicit clipping of species between 0 and 1.

define/models/species/species-transport-expert/Linearize higher order convection sources?

Enables/disables linearization of higher order convection sources.

define/models/species/species-transport-expert/Linearizesecondary diffusion sources?

Enables/disables linearization of species secondary diffusion sources.

define/models/species/species-transport-expert/Enable mesh quality based first-second order species blending?

Enables/disables skewness-based blending of first and second order convection.

define/models/species/species-transport-expert/Minimum cell quality threshold for blending

Specifies the minimum cell orthogonal quality for blending.

define/models/species/species-transport?

Enables/disables the species transport model.

define/models/species/stiff-chemistry?

Enables/disables stiff chemistry option.

define/models/species/surf-reaction-aggressiveness-factor?

Sets the surface reaction aggressiveness factor.

define/models/species/surf-reaction-netm-params

Sets the surface reaction parameters for the Non-Equilibrium Thermal Model.

define/models/species/thermal-diffusion?

Enables/disables thermal diffusion.

define/models/species/thickened-flame-model?

Enables/disables the Relaxation to Chemical Equilibrium model

define/models/species/volumetric-reactions?

Enables/disables volumetric reactions.

define/models/species/wall-surface-reactions?

Enables/disables wall surface reactions.

define/models/steady?

Enables/disables the steady solution model.

define/models/structure/

Enters the structure model menu.

define/models/structure/controls

Enters the structure controls menu.

define/models/structure/controls/amg-stabilization

Sets the algebraic multigrid (AMG) stabilization method for the structural model calculations.

define/models/structure/controls/max-iter

Sets the maximum number of iterations for the structural model calculations.

define/models/structure/controls/numerical-damping-factor?

Sets the damping factor for the structural model (that is, the amplitude decay factor γ in Equation 17.16 in the Theory Guide).

define/models/structure/expert/

Enters the structure expert menu.

define/models/structure/expert/explicit-fsi-force?

Enables/disables an explicit fluid-structure interaction force.

define/models/structure/expert/include-pop-in-fsi-force?

Enables/disables the inclusion of operating pressure into the fluid-structure interaction force.

define/models/structure/expert/include-viscous-fsi-force?

Enables/disables the inclusion of a viscous fluid-structure interaction force.

define/models/structure/linear-elasticity?

Enables the linear elasticity model.

define/models/structure/structure-off?

Disables the structural model.

define/models/swirl?

Enables/disables axisymmetric swirl velocity.

define/models/system-coupling-settings/

Enters the system coupling menu.

define/models/system-coupling-settings/use-face-or-element-based-data-transfer?

Answering `yes` at the prompt enables Fluent to use element data for mapping surface conservative quantities such as surface forces and heat flows.

define/models/two-temperature

Enters the Two-Temperature model menu.

define/models/two-temperature/enable?

Enables/disables the Two-Temperature model.

define/models/two-temperature/set-verbosity?

Specifies the level of detail printed in the console about the Two-Temperature model. For a verbosity of one, Fluent will print the number of cells that reach the temperature limit, have an excessive temperature change, or get a negative temperature.

define/models/unsteady-1st-order?

Selects the first-order implicit formulation for transient simulations.

define/models/unsteady-2nd-order-bounded?

Selects the bounded second-order implicit formulation for transient simulations.

define/models/unsteady-2nd-order?

Selects the second-order implicit formulation for transient simulations.

define/models/unsteady-global-time?

Selects the explicit transient formulation. This text command is only available for unsteady cases that use the density-based solver with the explicit formulation.

define/models/unsteady-structure-euler?

Selects the backward Euler method for the direct time integration of the finite element semi-discrete equation of motion. This text command is only available for transient simulations that use the structural model.

define/models/unsteady-structure-newmark?

Selects the Newmark method for the direct time integration of the finite element semi-discrete equation of motion. This text command is only available for transient simulations that use the structural model.

define/models/viscous/

Enters the viscous model menu.

define/models/viscous/add-transition-model

Sets Transition model to account for transitional effects. The default is `none`, however you can select `gamma-algebraic` or `gamma-transport-eqn`.

After a transition model has been enabled, you have additional options in the submenu `transition-model-options`.

define/models/viscous/corner-flow-correction-ccorner

Sets the strength of the quadratic term of the corner flow correction. The default value is 1. This is available after the `corner-flow-correction?` option is enabled.

define/models/viscous/corner-flow-correction?

Enables/disables the corner flow correction.

define/models/viscous/curvature-correction-ccurv

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

define/models/viscous/curvature-correction?

Enables/disables the curvature correction.

`define/models/viscous/des-limiter-option`

Selects the DES limiter option (none, F1, F2, Delayed DES, or Improved Delayed DES).

`define/models/viscous/geko-options/`

Enters the GEKO model menu..

`define/models/viscous/geko-options/csep`

Sets `csep`, the parameter to optimize flow separation from smooth surfaces.

`define/models/viscous/geko-options/cnw`

Sets `cnw`, the parameter to optimize flow in non-equilibrium near wall regions.

`define/models/viscous/geko-options/cmix`

Sets `cmix`, the parameter to optimize strength of mixing in free shear flows.

`define/models/viscous/geko-options/cjet`

Sets `cjet`, the parameter to optimize free shear layer mixing (optimize free jets independent of mixing layer).

`define/models/viscous/geko-options/blending-function`

Sets the blending function, which deactivates `cmix` and `cjet` inside boundary layers.

`define/models/viscous/geko-options/creal`

Sets `creal`, a realizability limiter that ensures positive normal stresses in the entire domain.

`define/models/viscous/geko-options/cnw_sub`

Sets `cnw_sub`, which allows the adjustment of log-layer and cf.

`define/models/viscous/geko-options/cjet_aux`

Sets `cjet_aux`, which allows fine-tuning of the parameter to optimize free jets.

`define/models/viscous/geko-options/cbf_tur`

Sets `cbf_tur`, a factor for the main blending factor, controlling the thickness of the layer near walls.

`define/models/viscous/geko-options/cbf_lam`

Sets `cbf_lam`, a part of the blending function and shields the laminar boundary layer.

`define/models/viscous/geko-options/geko-defaults`

Restores the defaults of all GEKO parameters.

define/models/viscous/geko-options/wall-distance-free?

Enables/disables the wall distance free version of the GEKO model.

define/models/viscous/detached-eddy-simulation?

Enables/disables detached eddy simulation.

define/models/viscous/inviscid?

Enables/disables inviscid flow model.

define/models/viscous/ke-easm?

Enables/disables the EASMk-ε turbulence model.

define/models/viscous/ke-realizable?

Enables/disables the realizable$k$-ε turbulence model.

define/models/viscous/ke-rng?

Enables/disables the RNGk-ε turbulence model.

define/models/viscous/ke-standard?

Enables/disables the standard$k$-ε turbulence model.

define/models/viscous/k-kl-w?

Enables/disables the k-kl-ω turbulence model.

define/models/viscous/kw-easm?

Enables/disables the EASMk-ω turbulence model.

define/models/viscous/kw-low-re-correction?

Enables/disables the$k$-ω low Re option.

define/models/viscous/kw-shear-correction?

Enables/disables the$k$-ω shear-flow correction option. This text command is only available for the standard k-ω model and the stress-omega RSM model.

define/models/viscous/kw-bsl?

Enables/disables the baseline (BSL)k-ω turbulence model.

define/models/viscous/kw-geko?

Enables/disables the generalized$k$-ω (GEKO) turbulence model.

352

`define/models/viscous/kw-sst?`

Enables/disables the SST k-ω turbulence model.

`define/models/viscous/kw-standard?`

Enables/disables the standard k-ω turbulence model.

`define/models/viscous/laminar?`

Enables/disables laminar flow model.

`define/models/viscous/large-eddy-simulation?`

Enables/disables large eddy simulation.

`define/models/viscous/les-dynamic-energy-flux?`

Enables/disables the dynamic sub-grid scale turbulent Prandtl Number.

`define/models/viscous/les-dynamic-scalar-flux?`

Enables/disables the dynamic sub-grid scale turbulent Schmidt Number.

`define/models/viscous/les-subgrid-dynamic-fvar?`

Enables/disables the dynamic subgrid-scale mixture fraction variance model.

`define/models/viscous/les-subgrid-smagorinsky?`

Enables/disables the Smagorinsky-Lilly subgrid-scale model.

`define/models/viscous/les-subgrid-tke?`

Enables/disables kinetic energy transport subgrid-scale model.

`define/models/viscous/les-subgrid-wale?`

Enables/disables WALE subgrid-scale model.

`define/models/viscous/les-subgrid-wmles-s_minus_omega?`

Enables/disables the WMLES S-Ω subgrid-scale model.

`define/models/viscous/les-subgrid-wmles?`

Enables/disables the WMLES subgrid-scale model.

`define/models/viscous/mixing-length?`

Enables/disables mixing-length (algebraic) turbulence model.

`define/models/viscous/multiphase-turbulence/`

Enters the multiphase turbulence menu.

`define/models/viscous/multiphase-turbulence/multiphase-options`

Enables/disables multiphase options.

`define/models/viscous/multiphase-turbulence/rsm-multiphase-models`

Selects Reynolds Stress multiphase model.

`define/models/viscous/multiphase-turbulence/turbulence-multiphase-models`

Selects k-ε multiphase model.

`define/models/viscous/near-wall-treatment/`

Enters the near wall treatment menu.

`define/models/viscous/near-wall-treatment/enhanced-wall-treatment?`

Enables/disables enhanced wall functions.

`define/models/viscous/near-wall-treatment/menter-lechner?`

Enables/disables the Menter-Lechner near-wall treatment.

`define/models/viscous/near-wall-treatment/non-equilibrium-wall-fn?`

Enables/disables non-equilibrium wall functions.

`define/models/viscous/near-wall-treatment/scalable-wall-functions?`

Enables/disables scalable wall functions.

`define/models/viscous/near-wall-treatment/standard-wall-fn?`

Enables/disables standard wall functions.

`define/models/viscous/near-wall-treatment/user-defined-wall-functions?`

Enables/disables user-defined wall functions.

`define/models/viscous/near-wall-treatment/werner-wengle-wall-fn?`

Enables/disables Werner-Wengle wall functions.

`define/models/viscous/near-wall-treatment/wf-pressure-gradient-effects?`

Enables/disables wall function pressure- gradient effects.

`define/models/viscous/near-wall-treatment/wf-thermal-effects?`

Enables/disables wall function thermal effects.

`define/models/viscous/reynolds-stress-model?`

Enables/disables the Reynolds-stress turbulence model.

define/models/viscous/rng-differential-visc?

Enables/disables the differential-viscosity model.

define/models/viscous/rng-swirl-model?

Enables/disables swirl corrections for rng-model.

define/models/viscous/rsm-bsl-based?

Enables/disables the stress-BSL Reynolds stress model.

define/models/viscous/rsm-linear-pressure-strain?

Enables/disables the linear pressure-strain model in RSM.

define/models/viscous/rsm-omega-based?

Enables/disables the stress-omega Reynolds stress model.

define/models/viscous/rsm-or-earsm-geko-option?

Enables/disables the RSM version of the GEKO model.

define/models/viscous/rsm-solve-tke?

Enables/disables the solution of T.K.E. in RSM model.

define/models/viscous/rsm-ssg-pressure-strain?

Enables/disables quadratic pressure-strain model in RSM.

define/models/viscous/rsm-wall-echo?

Enables/disables wall-echo effects in RSM model.

define/models/viscous/sa-alternate-prod?

Enables/disables strain/vorticity production in Spalart-Allmaras model.

define/models/viscous/sa-damping?

Enables/disables 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?`.

define/models/viscous/sa-enhanced-wall-treatment?

Enables/disables 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.

define/models/viscous/sas?

Enables/disables Scale-Adaptive Simulation (SAS) in combination with the SST k-ω turbulence model.

define/models/viscous/spalart-allmaras?

Enables/disables Spalart-Allmaras turbulence model.

define/models/viscous/transition-model-options/

Enters the transition model options menu.

define/models/viscous/transition-model-options/capg-hightu

Sets the algebraic transition model coefficient CAPG_HIGHTU.

define/models/viscous/transition-model-options/capg-lowtu

Sets the algebraic transition model coefficient CAPG_LOWTU.

define/models/viscous/transition-model-options/cbubble-c1

Sets the algebraic transition model coefficient CBUBBLE_C1.

define/models/viscous/transition-model-options/cbubble-c2

Sets the algebraic transition model coefficient CBUBBLE_C2.

define/models/viscous/transition-model-options/cfpg-hightu

Sets the algebraic transition model coefficient CFPG_HIGHTU.

define/models/viscous/transition-model-options/cfpg-lowtu

Sets the algebraic transition model coefficient CFPG_LOWTU.

define/models/viscous/transition-model-options/clambda-scale

Sets the algebraic transition model coefficient CLAMBDA_SCALE.

define/models/viscous/transition-model-options/critical-reynolds-number-correlation

Sets the critical Reynolds number correlation.

define/models/viscous/transition-model-options/ctu-hightu

Sets the algebraic transition model coefficient CTU_HIGHTU.

define/models/viscous/transition-model-options/ctu-lowtu

Sets the algebraic transition model coefficient CTU_LOWTU.

define/models/viscous/transition-model-options/rec-c1

Sets the algebraic transition model coefficient REC_C1.

define/models/viscous/transition-model-options/rec-c2

Sets the algebraic transition model coefficient REC_C2.

define/models/viscous/transition-model-options/rec-max

Sets the algebraic transition model coefficient REC_MAX.

define/models/viscous/transition-model-options/rv1-switch

Sets the algebraic transition model coefficient RV1_SWITCH.

define/models/viscous/transition-model-options/crossflow-transition?

Enables/disables the effects of crossflow instability.

define/models/viscous/transition-sst?

Enables/disables the transition SST turbulence model.

define/models/viscous/trans-sst-roughness-correlation?

Enables/disables the Transition-SST roughness correlation option.

define/models/viscous/turb-buoyancy-effects?

Enables/disables effects of buoyancy on turbulence.

define/models/viscous/turb-compressibility?

Enables/disables the compressibility correction option.

define/models/viscous/turbulence-expert/

Enters the turbulence expert menu.

define/models/viscous/turbulence-expert/kato-launder-model?

Enables/disables Kato-Launder modification.

define/models/viscous/turbulence-expert/kw-add-des?

Enables/disables Detached Eddy Simulation (DES) in combination with the currently selected BSL k-ω model or transition SST model. This text command is only available for transient cases.

define/models/viscous/turbulence-expert/kw-add-sas?

Enables/disables Scale-Adaptive Simulation (SAS) in combination with the currently selected ω-based URANS turbulence model. This text command is only available for transient cases.

`define/models/viscous/turbulence-expert/kw-vorticity-based-production?`

Enables/disables vorticity-based production.

`define/models/viscous/turbulence-expert/low-re-ke?`

Enables/disables the low-Re k-ε turbulence model.

`define/models/viscous/turbulence-expert/low-re-ke-index`

Specifies 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.

`define/models/viscous/turbulence-expert/production-limiter?`

Enables/disables Production Limiter modification.

`define/models/viscous/turbulence-expert/non-newtonian-modification?`

Enables/disables non-Newtonian modification for Lam-Bremhorst model.

`define/models/viscous/turbulence-expert/restore-sst-v61?`

Enables/disables SST formulation of v6.1.

`define/models/viscous/turbulence-expert/rke-cmu-rotation-term?`

Modifies 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.

`define/models/viscous/turbulence-expert/sbes-sdes-hybrid-model`

Selects the hybrid model, to specify whether you want to apply the Shielded Detached Eddy Simulation (SDES) model, Stress-Blended Eddy Simulation (SBES), or SBES with a user-defined function.

define/models/viscous/turbulence-expert/sbes-sgs-option

Selects the subgrid-scale model for the LES portion of your Stress-Blended Eddy Simulation (SBES).

define/models/viscous/turbulence-expert/sbes-update-interval-k-omega

Sets the number of time steps between updates of the k-ω part of the SBES model.

define/models/viscous/turbulence-expert/thermal-p-function?

Enables/disables Jayatilleke P function.

define/models/viscous/turbulence-expert/turb-add-sbes-sdes?

Enables/disables the Stress-Blended Eddy Simulation (SBES) model or Shielded Detached Eddy Simulation (SDES) model.

define/models/viscous/turbulence-expert/turb-non-newtonian?

Enables/disables turbulence for non-Newtonian fluids.

define/models/viscous/turbulence-expert/turbulence-damping?

Enables/disables turbulence damping and sets turbulence damping parameters.

define/models/viscous/turbulence-expert/turb-pk-compressible?

Enables/disables turbulent production due to compressible divergence.

define/models/viscous/user-defined

Selects user-defined functions to define the turbulent viscosity and the turbulent Prandtl and Schmidt numbers.

define/models/viscous/user-defined-transition

Sets user-defined transition correlations.

define/models/viscous/v2f?

Enables/disables V2F turbulence model.

define/models/viscous/zero-equation-hvac?

Enables/disables zero-equation HVAC turbulence model.

define/named-expressions/

Enters the named expressions menu.

define/named-expressions/add

Creates a new named expression.

define/named-expressions/compute

Computes and prints the value of an expression. This is only available for expressions that evaluate to a single value.

define/named-expressions/copy

Allows you to copy an existing named expression.

define/named-expressions/delete

Allows you to delete a named expression, as long as it is not in use.

define/named-expressions/export-to-tsv

Export one or more named expressions to a file in TSV format.

define/named-expressions/import-from-tsv

Import one or more named expressions saved in TSV format.

define/named-expressions/edit

Allows you to edit the definition of a named expression.

define/named-expressions/list

Lists all of the currently defined named expressions.

define/named-expressions/list-properties

Lists the definition of a named expression.

define/operating-conditions/

Enters the define operating conditions menu.

define/operating-conditions/gravity

Sets gravitational acceleration.

define/operating-conditions/gravity-mrf-rotation

Enables/disables rotation of gravity vector in moving reference frame simulations. If enabled, the gravity vector will rotate with respect to the moving reference frame such that the direction of gravity in global coordinates remains fixed.

define/operating-conditions/operating-density?

Enables/disables use of a specified operating density. This command is available only for single-phase flows.

define/operating-conditions/operating-density/

Enters the operating density menu. This menu is available only for multiphase flows.

define/operating-conditions/operating-density/print-operating-density

Prints the operating density value.

define/operating-conditions/operating-density/set-operating-density

Allows you to select the operating density method in your simulation. You can choose from:
- `0 = minimum-phase-averaged`
- `1 = primary-phase-averaged`
- `2 = mixture-averaged`
- `3 = user-input`

See Modeling Buoyancy-Driven Multiphase Flow in the *Fluent User's Guide* for more information about these options.

define/operating-conditions/operating-pressure

Sets the operating pressure.

define/operating-conditions/operating-temperature

Sets the operating temperature for Boussinesq.

define/operating-conditions/reference-pressure-location

Sets a location that determines the reference pressure cell or cells (depending on the selected reference pressure method). The pressure value in such cells can be used to adjust the gauge pressure field after each iteration to keep it from floating.

define/operating-conditions/reference-pressure-method

Specifies the method used for adjusting the gauge pressure field after each iteration to keep it from floating: the method can assume that all of the cell zones are connected by internal boundary zones (such as interior zones), or can account for connected and disconnected cell zones. Such adjustment is only allowed for incompressible flows, and will be limited partially or entirely by the presence of a pressure boundary.

define/operating-conditions/set-state

Selects state for real gas EOS subcritical condition.

define/operating-conditions/used-ref-pressure-location

Prints the coordinates of the reference pressure cell or cells (depending on the selected reference pressure method). The pressure value in such cells can be used to adjust the gauge pressure field after each iteration to keep it from floating.

define/operating-conditions/use-inlet-temperature-for-operating-density

Uses inlet temperature to calculate operating density.

define/overset-interfaces/

Enters the overset interfaces menu.

define/overset-interfaces/adapt/

Enters the overset adaption menu.

define/overset-interfaces/adapt/adapt-mesh

Marks and adapts the mesh to improve your overset mesh.

define/overset-interfaces/adapt/mark-adaption

Marks cells to identify those that can be adapted in order to improve your overset mesh.

define/overset-interfaces/adapt/set/

Enters the overset adaption set menu.

define/overset-interfaces/adapt/set/adaption-sweeps

Sets the number of rounds of adaption applied during each adaption cycle.

define/overset-interfaces/adapt/set/automatic-adaption?

Enables/disables the option to adapt overset meshes automatically during solution update, to remove orphans and/or reduce size mismatches between donor and receptor cells.

define/overset-interfaces/adapt/set/buffer-layers

Sets the number of cell layers marked in addition to the cells marked for orphan adaption.

define/overset-interfaces/adapt/set/length-ratio-max

Sets the length scale ratio threshold used to determine which cells are marked for adaption based on donor-receptor cell size differences.

define/overset-interfaces/adapt/set/mark-coarsening?

Enables/disables the option to coarsen the mesh if mesh refinement is no longer needed. This option is enabled by default.

define/overset-interfaces/adapt/set/mark-fixed-orphans?

Enables/disables the option to also adapt based on cells that are not actual orphans because they were fixed by accepting neighbor donors. This option is only applied if `define/overset-interfaces/adapt/set/mark-orphans?` is enabled.

define/overset-interfaces/adapt/set/mark-orphans?

Enables/disables the option to adapt for orphan reduction.

define/overset-interfaces/adapt/set/mark-size?

Enables/disables the option to adapt based on donor-receptor cell size differences.

define/overset-interfaces/adapt/set/maximum-refinement-level

Sets the maximum level of refinement during overset adaption, in conjunction with the value set using the `mesh/adapt/set/maximum-refinement-level` text command (the larger of the two values is used).

define/overset-interfaces/check

Checks the integrity of the overset interfaces. Reports orphan cells and errors in the domain connectivity.

define/overset-interfaces/clear

Clears the domain connectivity of an overset interface. This text command is only available when `define/overset-interfaces/options/expert?` is set to `yes`.

define/overset-interfaces/clear-all

Clears the domain connectivity of all overset interfaces. This text command is only available when `define/overset-interfaces/options/expert?` is set to `yes`.

define/overset-interfaces/create

Creates an overset interface.

define/overset-interfaces/cut-control/

Enters the overset hole cut control menu.

define/overset-interfaces/cut-control/add

Adds a hole cut control, so that a boundary zone does not cut specified cell zones.

define/overset-interfaces/cut-control/cut-seeds/

Enters the overset hole cut seeds menu.

define/overset-interfaces/cut-control/cut-seeds/cut-seeds-for-all-component-zones?

Specifies whether all component zones get cut seeds, which cut pilot holes into the overlapping meshes that will then be enlarged during overlap minimization.

define/overset-interfaces/cut-control/delete

Deletes the hole cut control for a boundary zone.

define/overset-interfaces/cut-control/delete-all

Deletes the hole cut controls for all boundary zones.

define/overset-interfaces/cut-control/list

Lists the defined hole cut controls.

define/overset-interfaces/debug-hole-cut

Debugging tool to troubleshoot hole cutting of overset interfaces. This text command is only available when `define/overset-interfaces/options/expert?` is set to `yes`.

define/overset-interfaces/delete

Deletes an overset interface.

define/overset-interfaces/delete-all

Deletes all overset interfaces in the domain.

define/overset-interfaces/display-cells

Displays the overset cells marked using the text command `define/overset-interfaces/mark-cells`.

define/overset-interfaces/grid-priorities

Allows you to specify grid priorities on background and component meshes, used in the overlap minimization of an overset interface.

define/overset-interfaces/intersect

Executes the hole cutting of an overset interface and establishes the domain connectivity. This text command is only available when `define/overset-interfaces/options/expert?` is set to `yes`.

define/overset-interfaces/intersect-all

Executes hole cutting for all overset interfaces in the domain. This text command is only available when `define/overset-interfaces/options/expert?` is set to `yes`.

define/overset-interfaces/list

Lists information about the overset interfaces. The output depends on the overset verbosity setting.

define/overset-interfaces/mark-cell-change

Marks cells that have undergone a specified overset cell type change (from `solve`,`receptor`, or `dead` to any other type) in the last time step. Adaption registers are automatically filled based on these markings. This text command is only available for unsteady simulations and if `define/overset-interfaces/options/expert?` is set to `yes`.

define/overset-interfaces/mark-cells

Marks the specified overset cells (solve, receptor, donor, orphan, or dead) and fills registers based on the markings. Display the marked cells via the text command `define/overset-interfaces/display-cells`.

define/overset-interfaces/options/

Enters the overset interface options menu.

define/overset-interfaces/options/auto-create?

Enables the automatic creation of a default overset interface during initialization or mesh motion update.

define/overset-interfaces/options/dead-cell-update?

Enables/disables the updating of dead cells during the solution process, which may be helpful for some moving and dynamic mesh simulations (though at the cost of solver performance). You can enter the number of layers of dead cells adjacent to the receptor cells that you want updated, or enter -1 to request that all dead cells are updated. This text command is only available for transient simulations when define/overset-interfaces/options/expert? is set to yes.

define/overset-interfaces/options/donor-priority-method

Allows you to specify whether the cell donor priority used in the overlap minimization of an overset interface is inversely proportional to either the cell size or the distance to the nearest boundary.

define/overset-interfaces/options/expert?

Enables / disables overset-related expert tools.

define/overset-interfaces/options/mesh-interfaces?

Allows the inclusion of non-conformal mesh interfaces inside overset cell zones, as long as the mesh interfaces do not spatially overlap with the cells where the overset interfaces lie.

define/overset-interfaces/options/minimize-overlap?

Allows you to disable overlap minimization during hole cutting.

define/overset-interfaces/options/modified-donor-search?

Enables / disables modified donor search parameters. When enabled, these parameters result in a more extended donor search, which can be helpful for meshes with orphans. This text command is only available when define/overset-interfaces/options/expert? is set to yes.

define/overset-interfaces/options/modified-hole-cutting?

Enables / disables modified hole cutting parameters. When enabled, the parameters used can help avoid dead cell zones that result from a hole cutting failure. This text command is only available when define/overset-interfaces/options/expert? is set to yes.

define/overset-interfaces/options/node-connected-donors?

Allows you to switch between face or node connected donor cells. This text command is only available when define/overset-interfaces/options/expert? is set to yes.

define/overset-interfaces/options/overlap-boundaries?

Allows you to disable the detection of overlapping boundaries during hole cutting, in order to reduce the computational expense for cases that do not include such boundaries. When enabled, this text command also allows you to specify which boundary zones should be considered when detecting overlapping boundaries; this can be helpful when the default hole cutting process fails.

define/overset-interfaces/options/parallel

Allows you to select the algorithm used for overset parallel processing. You can select from the default cell exchange model or the local cell zone replication model; the cell exchange model should require less memory and may result in faster calculations.

define/overset-interfaces/options/partial-cut-faces?

Enables/disables enhanced hole cutting, where partially overlapping cut faces are decomposed into overlapping and non-overlapping fractions, and only the non-overlapping fractions participate in the hole cutting. This can help avoid the situation where a region or cell zone is erroneously identified as dead because of leakage during flood filling, even though it has overlapping boundaries that match well. When this text command is disabled, the partially overlapping faces are entirely protected from cutting.

define/overset-interfaces/options/render-receptor-cells?

Allows you to enable visualization of receptor cells in contour and mesh displays.

define/overset-interfaces/options/solve-island-removal

Sets the method used to control the removal of isolated patches of solve cells. This text command is only available when `define/overset-interfaces/options/expert?` is set to `yes`.

define/overset-interfaces/options/transient-caching

Sets the options to control caching of entities in transient overset simulations. This text command is only available when `define/overset-interfaces/options/expert?` is set to `yes`.

define/overset-interfaces/options/update-before-case-write?

Enables/disables the updating of the overset interfaces before writing a case file in the Common Fluids Format (CFF). This text command is only available when `define/overset-interfaces/options/expert?` is set to `yes`.

define/overset-interfaces/options/verbosity

Specifies the level of detail printed in the console about overset interfaces.

define/overset-interfaces/set-mark-bounds

Allows you to set bounds so that any marking of cells is performed within a spherical region based on a specified origin and radius. This can be useful when diagnosing a local problem on a large mesh. This text command is only available if `define/overset-interfaces/op-`

tions/expert? is set to yes. After defining the bounds, you can then mark the cells using the define/overset-interfaces/mark-cell-change or define/overset-inter-faces/mark-cells text command. To disable the bounds, enter a radius of 0.

define/parameters/

Enters the parameters menu.

define/parameters/**enable-in-TUI?**

Enables/disables parameters in the text user interface.

define/parameters/**input-parameters/**

Enters the input-parameters menu.

define/parameters/input-parameters/**delete**

Deletes an input parameter.

define/parameters/input-parameters/**edit**

Edits an input parameter.

define/parameters/**output-parameters/**

Enters the output-parameters menu.

define/parameters/output-parameters/**create**

Creates an output parameter.

define/parameters/output-parameters/**delete**

Deletes an output parameter.

define/parameters/output-parameters/**edit**

Edits an output parameter.

define/parameters/output-parameters/**print-all-to-console**

Displays all parameter values in the console.

define/parameters/output-parameters/**print-to-console**

Displays parameter value in the console.

define/parameters/output-parameters/**rename**

Renames an output parameter.

define/parameters/output-parameters/**write-all-to-file**

Writes all parameter values to file.

define/parameters/output-parameters/write-to-file

Writes parameter value to file.

define/**periodic-conditions/**

Enters the periodic conditions menu.

define/periodic-conditions/**massflow-rate-specification?**

Enables/disables specification of mass flow rate at the periodic boundary.

define/periodic-conditions/**pressure-gradient-specification?**

Enables/disables specification of pressure gradient at the periodic boundary.

define/**phases/**

Enters the phases menu.

define/phases/**iac-expert/**

Enters the IAC expert setting menu.

define/phases/iac-expert/**hibiki-ishii-model**

Sets Hibiki-Ishii model coefficients.

define/phases/iac-expert/**ishii-kim-model**

Sets Ishii-Kim model coefficients.

define/phases/iac-expert/**yao-morel-model**

Sets Yao-Morel model coefficients.

define/phases/**set-domain-properties/**

Enters the menu to set phase domain properties.

define/phases/set-domain-properties/**change-phases-names?**

Allows you to change the names of all the phases in your simulation.

define/phases/set-domain-properties/**interaction-domain/**

Enters the menu to set the interaction domain properties.

define/phases/set-domain-properties/interaction-domain/**forces/**

Enters the menu to set interfacial forces models.

define/phases/set-domain-properties/interaction-domain/forces/**drag**

Specifies the drag function, drag modification, and drag factor for each pair of phases. This command is available only with the Eulerian and Mixture multiphase models.

define/phases/set-domain-properties/interaction-domain/forces/lift/

Enters the menu to set the lift force. This item is available only with the Eulerian multiphase model.

define/phases/set-domain-properties/interaction-domain/forces/lift/lift

Specifies the lift function for each pair of phases.

define/phases/set-domain-properties/interaction-domain/forces/lift/lift-shaver-podowski?

Enables/disables the Shaver-Podowski lift correction.

define/phases/set-domain-properties/interaction-domain/forces/restitution

Specifies the restitution coefficient for collisions between each pair of granular phases, and for collisions between particles of the same granular phase. This command is available only for multiphase flows with two or more granular phases.

define/phases/set-domain-properties/interaction-domain/forces/slip-velocity

Specifies the slip velocity function for each secondary phase with respect to the primary phase. This command is available only for the Mixture multiphase model.

define/phases/set-domain-properties/interaction-domain/forces/surface-tension/

Enters the menu to set surface tension models.

define/phases/set-domain-properties/interaction-domain/forces/surface-tension/jump-adhesion?

Enables/disables the treatment of the contact angle specification at the porous jump boundary. This command is available only for the VOF multiphase model with the continuum surface stress model option and when `sfc-modeling?` is enabled.

define/phases/set-domain-properties/interaction-domain/forces/surface-tension/sfc-modeling?

Allows you to include the effects of surface tension along the fluid-fluid interface. This option is only available for the VOF and Eulerian multiphase models.

define/phases/set-domain-properties/interaction-domain/forces/surface-tension/sfc-model-type

Selects the surface tension model. You can choose between the continuum surface force and continuum surface stress methods. This item is available only when `sfc-modeling?` is enabled.

define/phases/set-domain-properties/interaction-domain/forces/surface-tension/sfc-tension-coeff

Specifies the surface tension coefficient for each pair of phases.

define/phases/set-domain-properties/interaction-domain/forces/surface-tension/wall-adhesion?

Enables/disables the specification for a wall adhesion angle. This item is available only when `sfc-modeling?` is enabled.

define/phases/set-domain-properties/interaction-domain/forces/turbulent-dispersion

Specifies the turbulent dispersion model for each primary-secondary phase pair. This command is available only with the Eulerian multiphase model.

define/phases/set-domain-properties/interaction-domain/forces/turbulence-interaction

Specifies the turbulence interaction model for each primary-secondary phase pair. This command is available only with the Eulerian multiphase model.

define/phases/set-domain-properties/interaction-domain/forces/virtual-mass/

Enters the menu to set virtual mass models.

define/phases/set-domain-properties/interaction-domain/forces/virtual-mass/virtual-mass?

Allows you to include the virtual mass force effect that occurs when a secondary phase accelerates relative to the primary phase. This command is available only with the Eulerian multiphase model.

define/phases/set-domain-properties/interaction-domain/forces/virtual-mass/vmass-coeff

Specifies the virtual mass coefficient for each pair of phases. This option is available only if `virtual-mass?` is enabled.

define/phases/set-domain-properties/interaction-domain/forces/virtual-mass/vmass-implicit?

Enables/disables the implicit method for the virtual mass force. This option can improve convergence in some cases. This option is available only if `virtual-mass?` is enabled.

define/phases/set-domain-properties/interaction-domain/forces/virtual-mass/vmass-implicit-options

Specifies what form of the implicit method to use (`default`, `option-2`, or `option-3`). `default` models the entire virtual mass force while `option-2`

and `option-3` model truncated expressions which may further improve convergence. This option is available only if `vmass-implicit?` is enabled.

define/phases/set-domain-properties/interaction-domain/forces/wall-lubrication

Specifies the wall lubrication model for each primary-secondary phase pair. This command is available only with the Eulerian multiphase model.

define/phases/set-domain-properties/interaction-domain/heat-mass-reactions/

Enters the menu to set heat, mass-transfer, and reaction models.

define/phases/set-domain-properties/interaction-domain/heat-mass-reactions/cavitation/

Enters the menu to set cavitation models. This option is available only for the Mixture multiphase model with the Singhal-et-al cavitation model enabled via `solve/set/advanced/singhal-et-al-cavitation-model`.

define/phases/set-domain-properties/interaction-domain/heat-mass-reactions/cavitation/cavitation?

Allows you to include the effects of cavitation. When the cavitation is enabled, you must specify vaporization pressure, surface tension coefficient, and non-condensable gas mass fraction using the `define/phases/set-domain-properties/interaction-domain/heat-mass-reactions/cavitation/caviation` text command. If multiple species are included in one or more secondary phases, or the heat transfer due to phase change is considered, the mass transfer mechanism must be defined before enabling the cavitation model.

define/phases/set-domain-properties/interaction-domain/heat-mass-reactions/cavitation/cavitation

Sets the vaporization pressure, surface tension coefficient, and non-condensable gas mass fraction. This command is available only when `cavitation?` is enabled.

define/phases/set-domain-properties/interaction-domain/heat-mass-reactions/heat-coeff

Species the heat transfer coefficient function between each pair of phases (`constant-htc`, `nusselt-number`, `ranz-marshall`, `hughmark`, `tomiyama`, `fixed-to-sat-temp`, `two-resistance`, or `user-defined`). This command is enable only with the Eulerian multiphase model.

define/phases/set-domain-properties/interaction-domain/heat-mass-reactions/mass-transfer

Sets the mass transfer mechanisms.

define/phases/set-domain-properties/interaction-domain/heat-mass-reactions/reactions

Allows you to define multiple heterogeneous reactions and stoichiometry. This option is available only with the species model.

define/phases/set-domain-properties/interaction-domain/interfacial-area/

Enters the menu to set interfacial area models. This menu is available only for the Mixture and Eulerian multiphase models.

define/phases/set-domain-properties/interaction-domain/interfacial-area/interfacial-area

Specifies the interfacial area model for each pair of phases.

define/phases/set-domain-properties/interaction-domain/model-transition/

Enters the menu to set model transition mechanisms.

define/phases/set-domain-properties/interaction-domain/model-transition/model-transition

Sets the VOF-to-DPM model transition mechanism.

define/phases/set-domain-properties/interaction-domain/numerics/

Enters the menu to set numerics models. This menu is available for multiphase models with the sharp-dispersed and phase localized discretization interface modeling options (set in `define/models/multiphase/interface-modeling-options`).

define/phases/set-domain-properties/interaction-domain/numerics/interphase-discretization/

Enters the menu to set interphase discretization models.

define/phases/set-domain-properties/interaction-domain/numerics/interphase-discretization/interphase-discr?

Enables/disables phase localized compressive scheme.

define/phases/set-domain-properties/interaction-domain/numerics/interphase-discretization/slope-limiter

Specifies the slope limiter to set a specific discretization scheme for each phase pair. A value of 0 corresponds to first order upwind, a value of 1 corresponds to second order upwind, a value of 2 applies the compressive scheme, and a value between 0 and 2 corresponds to a blended scheme. This option is available only when `interphase-discr?` is enabled.

define/phases/set-domain-properties/phase-domains/

Enters the menu to select a specific phase.

define/phases/set-domain-properties/phase-domains/*phasename*/

Enters the menu for setting properties for the selected phase *phasename*, where *phasename* is the name of the phase you selected using the `phase-domains/` text command.

define/phases/set-domain-properties/phase-domains/*phasename*/diameter

Sets the diameter of the particles of the selected phase *phasename*. This text command is available for secondary phases only.

define/phases/set-domain-properties/phase-domains/*phasename*/granular-properties/

Enters the menu for setting properties for the granular phase. This text command is available only when the `granular?` text command is enabled.

define/phases/set-domain-properties/phase-domains/*phasename*/granular-properties/bulk-viscosity

Sets the solids bulk viscosity (constant, lun-et-al, or user-defined).

define/phases/set-domain-properties/phase-domains/*phasename*/granular-properties/elasticity-modulus

Specifies the elasticity modulus (derived or user-defined).

define/phases/set-domain-properties/phase-domains/*phasename*/granular-properties/friction-angle

Sets the angle of internal friction (constant or user-defined). This text command is available only when `schaeffer` is selected as the friction viscosity model.

define/phases/set-domain-properties/phase-domains/*phasename*/granular-properties/friction-packing

Sets a threshold volume fraction at which the frictional regime becomes dominant (constant or user-defined). The default value is 0.61. This text command is available only when `schaeffer` is selected as the friction viscosity model.

define/phases/set-domain-properties/phase-domains/*phasename*/granular-properties/friction-viscosity

Specifies a shear viscosity based on the viscous-plastic flow (none, constant, schaeffer, or user-defined). By default, the frictional viscosity is neglected (`none`).

> **Note:**
>
> When `schaeffer` is selected as the friction viscosity model, you must specify `friction-angle`, `friction-packing`, `frictional-modulus`, and `frictional-pressure`.

define/phases/set-domain-properties/phase-domains/*phasename*/granular-properties/frictional-modulus

Sets the frictional modulus (derived or user-defined). This text command is available only when schaeffer is selected as the friction viscosity model.

define/phases/set-domain-properties/phase-domains/*phasename*/granular-properties/frictional-pressure

Sets the pressure gradient term in the granular-phase momentum equation (none, johnson-et-al, syamlal-et-al, based-ktgf, or user-defined). This text command is available only when schaeffer is selected as the friction viscosity model.

define/phases/set-domain-properties/phase-domains/*phasename*/granular-properties/granular-conductivity

Sets temperature for the solids phase (constant, syamlal-obrien, gidaspow, or user-defined). This text command is available only if the pde-granular? text command is enabled.

define/phases/set-domain-properties/phase-domains/*phasename*/granular-properties/granular-temperature

Sets temperature for the solids phase (algebraic, constant, dpm-averaged, or user-defined). The dpm-averaged option is available only when using the Dense Discrete Phase Model (DDPM). This text command is available only if the pde-granular? text command is disabled.

define/phases/set-domain-properties/phase-domains/*phasename*/granular-properties/packed-bed?

Enables/disables the packed bed model.

define/phases/set-domain-properties/phase-domains/*phasename*/granular-properties/packing

Sets the maximum volume fraction for the granular phase (constant or user-defined). For monodispersed spheres the packing limit is about 0.63.

define/phases/set-domain-properties/phase-domains/*phasename*/granular-properties/pde-granular?

Enables/disables the partial differential equation (PDE) model for granular temperature. When the PDE model is enabled, you must specify granular-conductivity. Otherwise, you must specify granular-temperature.

define/phases/set-domain-properties/phase-domains/*phasename*/granular-properties/radial-distribution

Sets a correction factor that modifies the probability of collisions between grains when the solid granular phase becomes dense (lun-et-al, syamlal-obrien, ma-ahmadi, arastoopour, or user-defined).

define/phases/set-domain-properties/phase-domains/*phasename*/gran-ular-properties/solids-pressure

Sets the pressure gradient term in the granular-phase momentum equation (lun-et-al, syamlal-obrien, ma-ahmadi, user-defined, or none).

define/phases/set-domain-properties/phase-domains/*phasename*/gran-ular-properties/viscosity

Sets the kinetic part of the granular viscosity of the particles (constant, syamlal-obrien, gidaspow, or user-defined).

define/phases/set-domain-properties/phase-domains/*phasename*/granular?

Enables/disables the granular approach for the solid phase. This text command is available only for secondary phases in a Eulerian multiphase case. This approach is not compatible with the interfacial area concentration approach. For more details, see Defining a Granular Secondary Phase in the *Fluent User's Guide*.

define/phases/set-domain-properties/phase-domains/*phasename*/iac-properties/

Enters the menu for setting model properties for the interfacial area concentration. This text command is available only when the `iac?` text command is enabled.

define/phases/set-domain-properties/phase-domains/*phasename*/iac-properties/iac-breakage-kernel

Sets the breakage kernel (none, constant, hibiki-ishii, ishii-kim, yao-morel, or user-defined).

define/phases/set-domain-properties/phase-domains/*phasename*/iac-properties/iac-coalescence-kernel

Sets the coalescence kernel (none, constant, hibiki-ishii, ishii-kim, yao-morel, or user-defined).

define/phases/set-domain-properties/phase-domains/*phasename*/iac-properties/iac-critical-weber

Sets the critical Weber number. This value is required if you selected `yao-morel` as the breakage kernel.

define/phases/set-domain-properties/phase-domains/*phasename*/iac-properties/iac-dissipation-rate

Allows you to choose the dissipation function (constant, wu-ishii-kim, fluent-ke, or user-defined).

define/phases/set-domain-properties/phase-domains/*phasename*/iac-properties/iac-hydraulic-diam

Sets the hydraulic diameter. This value is required when you selected `wu-ishii-kim` for the IAC dissipation rate function.

define/phases/set-domain-properties/phase-domains/*phasename*/iac-properties/iac-max-diameter

Sets the maximum for the bubble diameters.

define/phases/set-domain-properties/phase-domains/*phasename*/iac-properties/iac-min-diameter

Sets the minimum for the bubble diameters.

define/phases/set-domain-properties/phase-domains/*phasename*/iac-properties/iac-nucleation-kernel

Sets the source term for the interfacial area concentration that models the rate of formation of the dispersed phase (none, constant, yao-morel, or user-defined).

define/phases/set-domain-properties/phase-domains/*phasename*/iac-properties/iac-surface-tension

Sets the attractive forces between the interfaces.

define/phases/set-domain-properties/phase-domains/*phasename*/iac?

Enables/disables the interfacial area concentration (IAC) approach. This text command is available for secondary phases only. The IAC approach is not compatible with the granular approach. See Defining the Interfacial Area Concentration via the Transport Equation and Defining the Interfacial Area Concentration in the *Fluent User's Guide* for details about using the IAC approach for the Mixture and Eulerian multiphase flows.

define/phases/set-domain-properties/phase-domains/*phasename*/material

Sets a material for the selected phase.

define/profiles/

Enters the boundary profiles menu.

define/profiles/delete

Deletes a profile.

define/profiles/delete-all

Deletes all boundary-profiles.

define/profiles/interpolation-method

Chooses the method for interpolation of profiles.

define/profiles/link-profile-to-reference-frame

Attaches a profile to a reference frame so that the profile will rotate according to the reference frame.

define/profiles/list-profiles

Lists all profiles.

define/profiles/list-profile-fields

Lists the fields of a particular profile.

define/profiles/morphing?

Enables/disables profile morphing options in Orient Profile panel.

define/profiles/update-interval

Sets interval between updates of dynamic profiles.

define/reference-frames/

Enters the reference frames menu.

define/reference-frames/add

Creates a new reference frame.

define/reference-frames/delete

Deletes the reference frame you specify.

define/reference-frames/display

Displays the reference frame you specify.

define/reference-frames/edit

Allows you to edit a reference frame.

define/reference-frames/hide

Removes the specified reference frame from the graphics window.

define/reference-frames/list

Lists all of the reference frames.

define/reference-frames/list-properties

Lists the properties of the reference frame you specify.

define/set-unit-system

Applies a standard set of units to all quantities. The options include `default`, `si`, `british`, and `cgs`.

define/solution-strategy/

Enters the automatic initialization and case modification strategy menu.

define/solution-strategy/automatic-case-modification/

Enters the automatic case modification menu.

define/solution-strategy/automatic-case-modification/before-init-modification

Specifies modification to be performed before initialization.

define/solution-strategy/automatic-case-modification/modifications

Specifies modifications to be performed during solution.

define/solution-strategy/automatic-case-modification/original-settings

Specifies modification to be performed after initialization to restore to original settings.

define/solution-strategy/automatic-initialization

Defines how the case is to be automatically initialized.

define/solution-strategy/continue-strategy-execution

Continues execution of the currently defined automatic initialization and case modification strategy.

define/solution-strategy/enable-strategy?

Enables/disables automatic initialization and case modification.

define/solution-strategy/execute-strategy

Executes the currently defined automatic initialization and case modification strategy.

define/turbo-model/

Enters the turbo menu.

define/turbo-model/blade-flutter-harmonics/

Enters the blade flutter harmonics options menu.

define/turbo-model/general-turbo-interface-settings/enable-harmonic-postprocessing?

Enables the computation of Fourier coefficients of pressure, velocity components, and temperature based on the blade flutter frequency of the simulation.

define/turbo-model/general-turbo-interface-settings/enable-harmonic-exports?

Enables the computation of real and imaginary harmonic pressure loads on user-defined surfaces based on the blade flutter frequency of the simulation.

define/turbo-model/general-turbo-interface-settings/write-harmonic-exports?

Writes a `.csv` file containing real and imaginary harmonic pressure loads that can be read into CFD-Post.

define/turbo-model/enable-turbo-model?

Enables/disables turbo model menu.

define/turbo-model/general-turbo-interface-settings/

Enters the General Turbo Interface options menu.

define/turbo-model/general-turbo-interface-settings/mixing-plane-model-settings/

Defines settings for the mixing plane model.

define/turbo-model/general-turbo-interface-settings/mixing-plane-model-settings/bands-type

Specifies the averaging bands for mixing as fixed or variable width.

define/turbo-model/general-turbo-interface-settings/mixing-plane-model-settings/formulation

Specifies whether the mixing plane model uses the intersector-based or side-boundary-based formulation.

define/turbo-model/general-turbo-interface-settings/mixing-plane-model-settings/number-of-bands

Sets the number of bands to be used for mixing.

define/turbo-model/general-turbo-interface-settings/mixing-plane-model-settings/number-of-inner-iterations

Sets the number of iterations used in averaging.

define/turbo-model/general-turbo-interface-settings/no-pitch-scale-model-settings/

Defines settings for the no pitch-scale model.

define/turbo-model/general-turbo-interface-settings/no-pitch-scale-model-settings/scale-mflux?

Scales mass flux to improve conservation.

define/turbo-model/general-turbo-interface-settings/pitch-scale-model-settings/

Defines settings for the pitch-scale model.

define/turbo-model/general-turbo-interface-settings/pitch-scale-model-settings/scale-mflux?

Scales mass flux to improve conservation.

define/turbo-model/number-of-blades-in-row/

Defines the number of blades in the turbo machine, used for calculating interblade phase angle.

define/turbo-model/turbo-create

Creates a general turbo interface.

define/turbo-model/turbo-topology/

Enters the turbo topology menu.

define/turbo-model/turbo-topology/define-topology

Defines a turbo topology.

define/turbo-model/turbo-topology/delete

Deletes a previously created turbo topology.

define/turbo-model/turbo-topology/mesh-method

Sets turbo structured mesh generation method.

define/turbo-model/turbo-topology/projection-method

Sets 2D projection method.

define/turbo-model/turbo-topology/search-method

Sets search method for a topology.

define/units

Sets unit conversion factors.

define/user-defined/

Enters the user-defined functions and scalars menu.

define/user-defined/1D-coupling

Loads 1D library.

define/user-defined/compiled-functions

Opens user-defined function library.

define/user-defined/execute-on-demand

Executes UDFs on demand.

define/user-defined/fan-model

Configures user-defined fan model.

define/user-defined/function-hooks

Hooks up user-defined functions.

define/user-defined/interpreted-functions

Loads interpreted user-defined functions.

define/user-defined/real-gas-models/

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

> **define/user-defined/real-gas-models/nist-multispecies-real-gas-model**
>
> Loads the NIST real-gas library.
>
> **define/user-defined/real-gas-models/nist-real-gas-model**
>
> Loads the NIST real-gas library.
>
> **define/user-defined/real-gas-models/nist-settings**
>
> Specifies the name and the location for the REFPROP library and fluid files.
>
> **define/user-defined/real-gas-models/set-state**
>
> Selects the state for NIST real gas model.
>
> **define/user-defined/real-gas-models/user-defined-multispecies-real-gas-model**
>
> Loads a user-defined multispecies real-gas library.
>
> **define/user-defined/real-gas-models/user-defined-real-gas-model**
>
> Loads the user-defined real-gas library.

define/user-defined/use-built-in-compiler?

Enables/disables the use of a built-in compiler (Clang) when the define/user-defined/compiled-functions text command is used. This text command / compiler is available for Windows only, and is provided as part of the Ansys Fluent installation. It is recommended that you enable this text command when the compiler you installed on your machine is an older version that is no longer supported. Note that the built-in compiler is used automatically if Fluent determines that you have not installed Microsoft Visual Studio or Clang on your computer, whether this text command is enabled or not.

define/user-defined/use-contributed-cpp?

Enables/disables use of the cpp utility included with the Ansys Fluent installation.

define/user-defined/user-defined-memory

Allocates user-defined memory.

define/user-defined/user-defined-scalars

Defines user-defined scalars.

Chapter 3: `display/`

`display/`**add-custom-vector**

Adds new custom vector definition.

`display/`**annotate**

Adds 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.

`display/`**clear-annotations**

Removes all annotations and attachment lines from the active graphics window.

`display/`**close-window**

Close a "user" graphics window. User windows are specified by number (1-50), with the associated number corresponding to the order, left-to-right, that the windows were created in.

`display/`**close-window-by-name**

Close a reserved graphics window (windows used for residuals, report plots, and animation definitions).

`display/`**display-states/**

Enter the display states menu.

`display/display-states/`**apply**

Apply a display state to the active graphics window.

`display/display-states/`**copy**

Copy the settings of an existing display state to another existing display state.

`display/display-states/`**create**

Create a new display state.

`display/display-states/`**delete**

Delete a display state.

`display/display-states/`**edit**

Edit a display state.

display/display-states/list

Print the names of the existing display states to the console.

display/display-states/read

Read in display states from a file.

display/display-states/use-active

Save the display state settings of the active graphics window to an existing display state. *This command is not available when the active window is displaying a 2D plot.*

display/display-states/write

Write one or more of the saved display states to a file.

display/**contour**

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

display/**display-custom-vector**

Displays custom vector.

display/**flamelet-data/**

Displays flamelet data.

display/flamelet-data/carpet-plot

Enables/disables display of carpet plot of a property.

display/flamelet-data/draw-number-box?

Enables/disables display of the numbers box.

display/flamelet-data/plot-1d-slice?

Enables/disables plot of the 1D-slice.

display/flamelet-data/write-to-file?

Enables/disables writing the 1D-slice to file instead of plot.

display/**graphics-window-layout**

Arranges the graphics window layout.

display/**hsf-file**

Loads an HSF file for viewing.

`display/`mesh

Displays 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`).

`display/`mesh-outline

Displays the mesh boundaries.

`display/`mesh-partition-boundary

Displays mesh partition boundaries.

`display/`multigrid-coarsening

Displays a coarse mesh level from the last multigrid coarsening.

`display/`objects/

Enters the graphics objects menu.

> **Important:**
>
> Numerical entries for graphics objects must be provided SI units (when applicable).

`display/objects/`add-to-graphics

Adds a contour, vector, pathline, particle track, scene, or mesh plot to the existing content in the graphics window.

`display/objects/`copy

Copies an existing contour, vector, pathline, particle track, scene, mesh, or XY plot definition.

`display/objects/`create

Creates a contour, vector, pathline, particle track, scene, mesh, or XY plot definition.

`display/objects/`delete

Deletes a contour, vector, pathline, particle track, scene, mesh, or XY plot definition.

`display/objects/`display

Displays a contour, vector, pathline, particle track, scene, mesh, or XY plot in the graphic windows replacing the existing content.

`display/objects/`edit

Edits a contour, vector, pathline, particle track, scene, or mesh plot definition.

`display/`open-window

Opens a graphics window.

`display/`particle-tracks/

Enters the particle tracks menu.

> **`display/particle-tracks/`particle-tracks**
>
> Calculates and displays particle tracks from defined injections.
>
> **`display/particle-tracks/`plot-write-xy-plot**
>
> Plots or writes an XY plot of particle tracks.

`display/`path-lines/

Enters the pathlines menu.

> **`display/path-lines/`path-lines**
>
> Displays pathlines from a surface.
>
> **`display/path-lines/`plot-write-xy-plot**
>
> Plots or writes an XY plot of pathlines.
>
> **`display/path-lines/`write-to-files**
>
> Writes pathlines to a file.

`display/`pdf-data/

Enters the PDF data menu.

> **`display/pdf-data/`carpet-plot**
>
> Enables/disables the display of a carpet plot of a property.
>
> **`display/pdf-data/`draw-number-box?**
>
> Enables/disables the display of the numbers box.
>
> **`display/pdf-data/`plot-1d-slice?**
>
> Enables/disables a plot of the 1D-slice.
>
> **`display/pdf-data/`write-to-file?**
>
> Enables/disables writing the 1D-slice to file instead of plot.

`display/`reacting-channel-curves

Plots the reacting channel variables.

`display/`profile

Displays profiles of a flow variable.

display/re-render

Re-renders the last contour, profile, or vector plot with updated surfaces, meshed, lights, colormap, rendering options, and so on, without recalculating the contour data.

display/re-scale

Re-renders the last contour, profile, or vector plot with updated scale, surfaces, meshes, lights, colormap, rendering options, and so on, but without recalculating the field data.

display/save-picture

Generates a "hardcopy" of the active window.

display/set/

Enters the set menu to set display parameters.

display/set/color-map/

Enters 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.

display/set/colors/

Enters the color options menu.

display/set/colors/automatic-skip

Specify whether the number of colormap labels is determined automatically. The default is yes.

display/set/colors/axis-faces

Sets the color of axisymmetric faces.

display/set/colors/background

Sets the background (window) color.

display/set/colors/color-by-type?

Determines whether to color meshes by type or by ID.

display/set/colors/far-field-faces

Sets the color of far field faces.

display/set/colors/free-surface-faces

Sets the color of free-surface faces.

`display/set/colors/`graphics-color-theme

Sets the color theme for the graphics window. The color options are for the background display, but changing the theme also changes the default colors for items that display in the graphics windows, like faces and edges.

`display/set/colors/`foreground

Sets the foreground (text, ruler, and window frame) color.

`display/set/colors/`highlight-color

Sets highlight color.

`display/set/colors/`inlet-faces

Sets the color of inlet faces.

`display/set/colors/`interface-faces

Sets the color of mesh interfaces.

`display/set/colors/`interior-faces

Sets the color of interior faces.

`display/set/colors/`internal-faces

Sets the color of internal interface faces.

`display/set/colors/`outlet-faces

Sets the color of outlet faces.

`display/set/colors/`periodic-faces

Sets the color of periodic faces.

`display/set/colors/`rans-les-interface-faces

Sets the color of RANS/LES interface faces.

`display/set/colors/`symmetry-faces

Sets the color of symmetric faces.

`display/set/colors/`traction-faces

Sets the color of traction faces.

`display/set/colors/`wall-faces

Sets the color of wall faces.

388

display/set/colors/list

    Lists available colors.

display/set/colors/reset-colors

    Resets individual mesh surface colors to the defaults.

display/set/colors/skip-label

    Sets the number of labels to be skipped in the colormap scale.

display/set/colors/surface

    Sets the color of surfaces.

display/set/contours/

    Enters the contour options menu.

display/set/contours/auto-range?

    Enables/disables auto-computation of the contour range.

display/set/contours/clip-to-range?

    Turns the clip to range option for filled contours on/off.

display/set/contours/coloring

    Specifies whether contours are displayed in bands or with smooth transitions. Note that you can only display smooth contours if node-values are enabled.

display/set/contours/contour-lines?

    With the `filled-contours?` and `node-values?` options enabled, this sets the use of lines to delineate the bands of color to coincide with the colormap.

display/set/contours/filled-contours?

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

display/set/contours/global-range?

    Turns the global range for contours on/off.

display/set/contours/line-contours?

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

display/set/contours/log-scale?

    Specifies a decimal or logarithmic color scale for contours.

display/set/contours/n-contour

Sets the number of contour levels.

display/set/contours/node-values?

Sets the option to use scalar field at nodes when computing the contours.

display/set/contours/render-mesh?

Determines whether or not to render the mesh on top of contours, vectors, and so on.

display/set/contours/surfaces

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

display/set/element-shrink

Sets 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.

display/set/filled-mesh?

Determines whether the meshes are drawn as wireframe or solid.

display/set/mesh-level

Sets coarse mesh level to be drawn.

display/set/mesh-partitions?

Enables/disables option to draw mesh partition boundaries.

display/set/mesh-surfaces

Sets surface IDs to be drawn as meshes. You can include a wildcard (*) within the surface names.

display/set/mesh-zones

Sets zone IDs to be drawn as meshes.

display/set/picture/

Enters the save-picture options menu.

display/set/picture/color-mode/

Enters the hardcopy/save-picture color mode menu.

display/set/picture/color-mode/color

Plots hardcopies in color.

display/set/picture/color-mode/gray-scale

Converts color to grayscale for hardcopy.

display/set/picture/color-mode/list

Displays the current hardcopy color mode.

display/set/picture/color-mode/mono-chrome

Converts color to monochrome (black and white) for hardcopy.

display/set/picture/x-resolution

Sets the width of raster-formatted images in pixels (0 implies current window size).

display/set/picture/y-resolution

Sets the height of raster-formatted images in pixels (0 implies current window size).

display/set/dpi

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

display/set/driver/

Enters the set hardcopy driver menu.

display/set/driver/dump-window

Sets the command used to dump the graphics window to a file.

display/set/driver/eps

Produces encapsulated PostScript (EPS) output for hardcopies.

display/set/driver/hsf

Produces HOOPS Visualize Stream Format (HSF) output for hardcopies.

display/set/driver/jpeg

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

display/set/driver/list

Lists the current hardcopy driver.

display/set/driver/options

Sets 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 hard-copy. The options may be combined by separating them with commas. The pen speed option is only meaningful to the HPGL driver.

display/set/driver/png

Uses PNG output for hardcopies.

display/set/driver/post-format/

Enters the PostScript driver format menu.

display/set/driver/post-format/fast-raster

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

display/set/driver/post-format/raster

Enables the standard raster file.

display/set/driver/post-format/rle-raster

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

display/set/driver/post-format/vector

Enables the standard vector file.

display/set/driver/post-script

Produces PostScript output for hardcopies.

display/set/driver/ppm

Produces PPM output for hardcopies.

display/set/driver/tiff

Produces TIFF output for hardcopies.

display/set/driver/vrml

Uses VRML output for hardcopies.

display/set/invert-background?

Exchanges foreground/background colors for hardcopy.

display/set/landscape?

Plots hardcopies in landscape or portrait orientation.

`display/set/`jpeg-hardcopy-quality

Controls the size and quality of how JPEG files are saved based on a scale of 0-100, with zero being low quality small files and 100 being high quality larger files.

`display/set/`preview

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

`display/set/`use-window-resolution?

Disables/enables the use of the current graphics window resolution when saving an image of the graphics window. If disabled, the resolution will be as specified for `x-resolution` and `y-resolution`.

`display/set/`lights/

Enters the lights menu.

`display/set/lights/`headlight-on?

Turns the light that moves with the camera on or off. This is controlled automatically by default.

`display/set/lights/`lighting-interpolation/

Sets lighting interpolation method.

`display/set/lights/lighting-interpolation/`automatic

Fluent automatically picks the best lighting method for the display in the graphics window.

`display/set/lights/lighting-interpolation/`flat

Uses flat shading for meshes and polygons.

`display/set/lights/lighting-interpolation/`gouraud

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

`display/set/lights/lighting-interpolation/`phong

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

`display/set/lights/`lights-on?

Turns all active lighting on/off. *This command is only available when the* `headlight-on?` *option is turned off (*`lights-on?` *is enabled when the headlight is on).*

`display/set/lights/`set-ambient-color

Sets the ambient color for the scene. The ambient color is the background light color in a scene.

`display/set/lights/`set-light

Adds or modifies a directional, colored light.

`display/set/`line-weight

Sets the line-weight factor for the window.

`display/set/`marker-size

Sets the size of markers used to represent points.

`display/set/`marker-symbol

Sets the type of markers used to represent points.

`display/set/`mesh-display-configuration

Changes the default mesh display. If set to**meshing**, it draws the mesh on edges and faces of the outline surfaces, colored by their zone ID with lighting enabled. If set to**solution**, it draws the mesh on edges and faces of the outline surfaces, colored by their zone type with lighting enabled. If set to **post-processing**, it draws the object outline with lighting disabled. If set to**classic**, it draws the mesh on all edges of the outline surfaces.

> **Note:**
>
> This only applies for 3D cases.

`display/set/`mirror-zones

Sets the zones about which the domain is mirrored (symmetry planes).

`display/set/`n-stream-func

Sets number of iterations used in computing stream function.

`display/set/`nodewt-based-interp?

Disables/enables the use of node weights for node-based gradients in postprocessing.

`display/set/`overlays?

Enables/disables overlays.

`display/set/`particle-tracks/

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

display/set/particle-tracks/arrow-scale

Sets the scale factor for arrows drawn on particle tracks.

display/set/particle-tracks/arrow-space

Sets the spacing factor for arrows drawn on particle tracks.

display/set/particle-tracks/coarsen-factor

Sets the coarsening factor for particle tracks.

display/set/particle-tracks/display?

Determines whether particle tracks shall be displayed or only tracked.

display/set/particle-tracks/filter-settings/

Sets filter for particle display.

> **display/set/particle-tracks/filter-settings/enable-filtering?**
>
> Specifies whether particle display is filtered.
>
> **display/set/particle-tracks/filter-settings/filter-variable**
>
> Selects a variable used for filtering of particles.
>
> **display/set/particle-tracks/filter-settings/inside?**
>
> Specifies whether filter variable must be inside min/max to be displayed (else outside min/max).
>
> **display/set/particle-tracks/filter-settings/maximum**
>
> Specifies the upper bound for the filter variable.
>
> **display/set/particle-tracks/filter-settings/minimum**
>
> Specifies the lower bound for the filter variable.

display/set/particle-tracks/history-filename

Specifies the name of the particle history file.

display/set/particle-tracks/line-width

Sets the width for particle track.

display/set/particle-tracks/marker-size

Sets the size of markers used to represent particle tracks.

display/set/particle-tracks/particle-skip

Specifies how many particle tracks should be displayed.

display/set/particle-tracks/radius

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

display/set/particle-tracks/report-to

Specifies the destination for the report (console, file, none).

display/set/particle-tracks/report-type

Sets the report type for particle tracks.

display/set/particle-tracks/report-variables

Sets the report variables.

display/set/particle-tracks/report-default-variables

Sets the report variables to default.

display/set/particle-tracks/sphere-attrib

Specifies the size and number of slices to be used in drawing spheres.

display/set/particle-tracks/sphere-settings/

Sets filter for particle display.

display/set/particle-tracks/sphere-settings/auto-range?

Specifies whether displayed spheres should include auto range of variable to size spheres.

display/set/particle-tracks/sphere-settings/diameter

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

display/set/particle-tracks/sphere-settings/maximum

Sets the maximum value of the sphere to be displayed.

display/set/particle-tracks/sphere-settings/minimum

Sets the minimum value of the sphere to be displayed.

display/set/particle-tracks/sphere-settings/scale-factor

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

display/set/particle-tracks/sphere-settings/size-variable

Selects a particle variable to size the spheres.

display/set/particle-tracks/sphere-settings/smooth-parameter

Specifies number of slices to be used in drawing spheres.

`display/set/particle-tracks/sphere-settings/vary-diameter?`

Specifies whether the spheres can vary with another variable.

`display/set/particle-tracks/style`

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

`display/set/particle-tracks/track-single-particle-stream?`

Specifies the stream ID to be tracked.

`display/set/particle-tracks/twist-factor`

Sets the scale factor for twisting (ribbons only).

`display/set/particle-tracks/vector-settings/`

Sets vector specific input.

`display/set/particle-tracks/vector-settings/color-variable?`

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

`display/set/particle-tracks/vector-settings/constant-color`

Specifies a constant color for the vectors.

`display/set/particle-tracks/vector-settings/length-to-head-ratio`

Specifies ratio of length to head for vectors and length to diameter for cylinders.

`display/set/particle-tracks/vector-settings/length-variable?`

Specifies whether the displayed vectors have length varying with another variable.

`display/set/particle-tracks/vector-settings/scale-factor`

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

`display/set/particle-tracks/vector-settings/style`

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

`display/set/particle-tracks/vector-settings/vector-length`

Specifies the length of constant vectors.

`display/set/particle-tracks/vector-settings/vector-length-variable`

Selects a particle variable to specify the length of vectors.

display/set/particle-tracks/vector-settings/vector-variable

Selects a particle vector function to specify vector direction.

display/set/path-lines/

Sets parameters for display of pathlines.

display/set/path-lines/arrow-scale

Sets the scale factor for arrows drawn on pathlines.

display/set/path-lines/arrow-space

Sets the spacing factor for arrows drawn on pathlines.

display/set/path-lines/display-steps

Sets the display stepping for pathlines.

display/set/path-lines/error-control?

Sets error control during pathline computation.

display/set/path-lines/line-width

Sets the width for pathlines.

display/set/path-lines/marker-size

Sets the marker size for particle drawing.

display/set/path-lines/maximum-error

Sets the maximum error allowed while computing the pathlines.

display/set/path-lines/maximum-steps

Sets the maximum number of steps to take for pathlines.

display/set/path-lines/radius

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

display/set/path-lines/relative-pathlines?

Enables/disables the tracking of pathlines in a relative coordinate system.

display/set/path-lines/reverse?

Sets direction of path tracking.

display/set/path-lines/sphere-attrib

Specifies the size and number of slices to be used in drawing spheres.

`display/set/path-lines/`step-size

Sets the step length between particle positions for pathlines.

`display/set/path-lines/`style

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

`display/set/path-lines/`time-step

Sets the time step between particle positions for pathlines.

`display/set/path-lines/`track-in-phase

Selects the phase in which particle pathlines will be computed (Multiphase Eulerian Model only).

`display/set/path-lines/`twist-factor

Sets the scale factor for twisting (ribbons only).

`display/set/`periodic-repeats

Sets number of periodic repetitions.

`display/set/`proximity-zones

Sets zones to be used for boundary cell distance and boundary proximity.

`display/set/`render-mesh?

Enables/disables rendering the mesh on top of contours, vectors, and so on.

`display/set/`rendering-options/

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

`display/set/rendering-options/`animation-option

Uses of`wireframe` or`all` during animation.

`display/set/rendering-options/`auto-spin?

Enables/disables mouse view rotations to continue to spin the display after the button is released.

`display/set/rendering-options/`color-map-alignment

Sets the color bar alignment.

`display/set/rendering-options/`device-info

Prints out information about your graphics driver.

`display/set/rendering-options/`double-buffering?

Enables/disables 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.

`display/set/rendering-options/`driver

Changes the current graphics driver. When enabling graphics display, you have various options: for Linux, the available drivers include `opengl` and `x11`; for Windows, the available drivers include `opengl`, `dx11` (for DirectX 11), and `msw` (for Microsoft Windows). You can also disable the graphics display window by entering `null`. For a comprehensive list of the drivers available to you, press the **Enter** key at the `driver>` prompt.

> **Note:**
>
> For any session that displays graphics in a graphics window and/or saves picture files, having the driver set to `x11`, `msw`, or `null` will cause the rendering / saving speed to be significantly slower.

`display/set/rendering-options/`face-displacement

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

`display/set/rendering-options/`hidden-line-method/

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

`display/set/rendering-options/hidden-line-method/`normal-hlr-algorithm

Normal hidden lines algorithm. This is the default.

`display/set/rendering-options/hidden-line-method/`mesh-display-hlr?

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

> **Note:**
>
> These commands (only available when **Graphics Color Theme** is set to **Black**) are deprecated and will be removed at a future release.

`display/set/rendering-options/hidden-lines?`

Turns hidden line removal on/off.

> **Note:**
>
> This command (only available when **Graphics Color Theme** is set to **Black**) is deprecated and will be removed at a future release.

`display/set/rendering-options/hidden-surfaces?`

Turns hidden surface removal on/off.

`display/set/rendering-options/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.

`display/set/rendering-options/hidden-surface-method/hardware-z-buffer`

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

`display/set/rendering-options/hidden-surface-method/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.

`display/set/rendering-options/hidden-surface-method/software-z-buffer`

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

`display/set/rendering-options/hidden-surface-method/z-sort-only`

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

`display/set/rendering-options/outer-face-cull?`

Enables/disables discarding outer faces during display.

`display/set/rendering-options/set-rendering-options`

Sets the rendering options.

`display/set/rendering-options/surface-edge-visibility`

Sets edge visibility flags for surfaces.

`display/set/reset-graphics`

Resets the graphics system.

display/set/title/

Sets problem title. This text only appears if the `display/set/windows/text/company?` text command is set to `yes` and if**Titles** is enabled in the **Display Options** dialog box.

display/set/title/left-top

Sets the title text for left top in title segment.

display/set/title/left-bottom

Sets the title text for left bottom in title segment.

display/set/title/right-top

Sets the title text for right top in title segment.

display/set/title/right-middle

Sets the title text for right middle in title segment.

display/set/title/right-bottom

Sets the title text for right bottom in title segment.

display/set/velocity-vectors/

Enters the menu to set parameters for display of velocity vectors.

display/set/velocity-vectors/auto-scale?

Auto-scales all vectors so that vector overlap is minimal.

display/set/velocity-vectors/color

Sets 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.

display/set/velocity-vectors/color-levels

Sets the number of colors used from the colormap.

display/set/velocity-vectors/component-x?

Sets the option to use only the$x$ component of the velocity vectors during display.

display/set/velocity-vectors/component-y?

Sets the option to use only the$y$ component of the velocity vectors during display.

display/set/velocity-vectors/component-z?

Sets the option to use only the$z$ component of the velocity vectors during display.

`display/set/velocity-vectors/`constant-length?

Sets the option to draw velocity vectors of constant length. This shows only the direction of the velocity vectors.

`display/set/velocity-vectors/`global-range?

Turns global range for vectors on/off.

`display/set/velocity-vectors/`in-plane?

Toggles the display of velocity vector components in the plane of the surface selected for display.

`display/set/velocity-vectors/`log-scale?

Toggles whether color scale is logarithmic or linear.

`display/set/velocity-vectors/`node-values?

Enables/disables the plotting of node values. Cell values will be plotted if "no".

`display/set/velocity-vectors/`relative?

Toggles the display of relative velocity vectors.

`display/set/velocity-vectors/`render-mesh?

Enables/disables rendering the mesh on top of contours, vectors, and so on.

`display/set/velocity-vectors/`scale

Sets the value by which the vector length will be scaled.

`display/set/velocity-vectors/`scale-head

Sets the value by which the vector head will be scaled.

`display/set/velocity-vectors/`style

Specifies the vector style that will be used when the vectors are displayed. You can choose from:**3d arrow**,**3d arrowhead**, **cone**,**filled-arrow**, **arrow**,**harpoon**, or **headless**.

`display/set/velocity-vectors/`surfaces

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

`display/set/`windows/

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

`display/set/windows/aspect-ratio`

Sets the aspect ratio of the active window.

`display/set/windows/axes/`

Enters the axes window options menu (3D only).

`display/set/windows/axes/border?`

Sets whether to draw a border around the axes window.

`display/set/windows/axes/bottom`

Sets the bottom boundary of the axes window.

`display/set/windows/axes/clear?`

Sets the transparency of the axes window.

`display/set/windows/axes/logo?`

Enables/disables visibility of the logo in graphics window.

`display/set/windows/axes/logo-color`

Sets logo color to white/black in graphics window.

`display/set/windows/axes/right`

Sets the right boundary of the axes window.

`display/set/windows/axes/visible?`

Turns axes visibility on/off.

`display/set/windows/main/`

Enters the main view window options menu.

`display/set/windows/main/border?`

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

`display/set/windows/main/bottom`

Sets the bottom boundary of the main viewing window.

`display/set/windows/main/left`

Sets the left boundary of the main viewing window.

`display/set/windows/main/right`

Sets the right boundary of the main viewing window.

display/set/windows/main/top

Sets the top boundary of the main viewing window.

display/set/windows/main/visible?

Turns visibility of the main viewing window on/off.

display/set/windows/ruler?

Turns the ruler on/off. Note that if you are running Fluent in 3D, then the view must be set to *orthographic*.

display/set/windows/scale/

Enters the color scale window options menu.

display/set/windows/scale/alignment

Sets the colormap position to the bottom, left, top, or right.

display/set/windows/scale/border?

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

display/set/windows/scale/bottom

Sets the bottom boundary of the color scale window.

display/set/windows/scale/clear?

Sets the transparency of the scale window.

display/set/windows/scale/format

Sets the number format of the color scale window. (for example, %0.2e)

display/set/windows/scale/font-size

Sets the font size of the color scale window.

display/set/windows/scale/left

Sets the left boundary of the color scale window.

display/set/windows/scale/margin

Sets the margin of the color scale window.

display/set/windows/scale/right

Sets the right boundary of the color scale window.

display/set/windows/scale/top

Sets the top boundary of the color scale window.

display/set/windows/scale/visible?

Turns visibility of the color scale window on/off.

display/set/windows/text/

Enters the text window options menu.

display/set/windows/text/application?

Shows/hides the application name in the picture.

display/set/windows/text/border?

Sets whether or not to draw a border around the text window.

display/set/windows/text/bottom

Sets the bottom boundary of the text window.

display/set/windows/text/clear?

Enables/disables text window transparency.

display/set/windows/text/company?

Enables/disables the display of your company name or other text defined using the `display/set/titles/` text command. The text appears in the title box. See Controlling the Titles, Axes, Ruler, Logo, and Colormap in the *Fluent User's Guide* for additional information.

display/set/windows/text/date?

Shows/hides the date in the picture.

display/set/windows/text/left

Sets the left boundary of the text window.

display/set/windows/text/right

Sets the right boundary of the text window.

display/set/windows/text/top

Sets the top boundary of the text window.

display/set/windows/text/visible?

Turns visibility of the text window on/off.

display/set/windows/video/

Enters the video window options menu.

display/set/windows/video/background

Sets 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.

display/set/windows/video/color-filter

Sets 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.

display/set/windows/video/foreground

Sets 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.

display/set/windows/video/on?

Enables/disables video picture settings.

display/set/windows/video/pixel-size

Sets the window size in pixels.

display/set/windows/xy/

Enters the XY plot window options menu.

display/set/windows/xy/border?

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

display/set/windows/xy/bottom

Sets the bottom boundary of the XY plot window.

display/set/windows/xy/left

Sets the left boundary of the XY plot window.

display/set/windows/xy/right

Sets the right boundary of the XY plot window.

display/set/windows/xy/top

Sets the top boundary of the XY plot window.

display/set/windows/xy/visible?

Turns visibility of the XY plot window on/off.

`display/set/zero-angle-dir`

Sets the vector having zero angular coordinates.

`display/set-list-tree-separator`

Sets the separator character for list tree.

`display/set-window`

Sets a "user" graphics window to be the active window. User windows are specified by number (1-50), with the associated number corresponding to the order, left-to-right, that the windows were created in.

`display/set-window-by-name`

Sets the specified graphics window as active. The graphics windows specified using this command are "reserved" windows, that is, residual monitors, report plots, and animation definitions. The name will match the name you provided when creating the object (report plots and animation definitions).

`display/surface/`

Enters the data surface-manipulation menu. For a description of the items in this menu, see sur-face/ (p. 517).

`display/surface-cells`

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

`display/surface-mesh`

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

`display/update-scene/`

Enters the scene options menu.

`display/update-scene/delete`

Deletes selected geometries.

`display/update-scene/display`

Displays selected geometries.

`display/update-scene/draw-frame?`

Enables/disables drawing the bounding frame.

`display/update-scene/iso-sweep`

Changes iso-sweep values.

display/update-scene/overlays?

Enables/disables the overlays option.

display/update-scene/pathline

Changes pathline attributes.

display/update-scene/select-geometry

Selects geometry to be updated.

display/update-scene/set-frame

Changes frame options.

display/update-scene/time

Changes time-step value.

display/update-scene/transform

Applies transformation matrix on selected geometries.

display/vector

Displays vectors of a space vector variable.

display/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.

display/view/

Enters the view manipulation menu. For a description of the items in this menu, see .

display/zone-mesh

Draws the mesh defined by specified face zones. Zone names can be indicated using wildcards (*).

Chapter 4: `exit / close-fluent`

`exit`

Exits program.

`close-fluent`

(Ansys Fluent in Workbench only) Exits program.

Chapter 5: `file/`

`file/async-optimize?`

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

`file/auto-save/`

Enters the auto save menu.

`file/auto-save/`**append-file-name-with**

Sets 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.

`file/auto-save/`**case-frequency**

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

`file/auto-save/`**data-frequency**

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

`file/auto-save/`**max-files**

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

`file/auto-save/`**overwrite-existing-files**

Overwrites existing files when files are automatically saved.

`file/auto-save/`**retain-most-recent-files**

Sets autosave to retain the 5 most recent files.

`file/auto-save/`**root-name**

Specifies the root name for the files that are saved.

`file/auto-save/`**save-data-file-every**

Specifies the type and frequency of the data file to be saved.

file/binary-files?

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

> **Important:**
>
> Common Fluids Format (CFF) Files are always saved in binary format (and therefore cannot be saved in ASCII format), so setting this command to 'no' has no effect on CFF files, only the legacy (`.cas` and `.dat`) format.

file/cff-files?

Specifies which format used to write mesh, case, and data files, and which format is prioritized when reading. Answering **yes** specifies that the Common Fluids Format (`.msh.h5`, `.cas.h5`, and `.dat.h5`) is used, whereas **no** specifies that the legacy format (`.msh`, `.cas`, and `.dat`) is used.

file/cffio-options/

Enters the cffio options menu, which provides options for the I/O of case and data files in the Common Fluids Format (CFF).

file/cffio-options/compression-level

Sets the compression level for CFF files. The compression level can be set between 0 and 9 with 0 being least compression (fastest) and 9 being highest compression (slowest).

file/cffio-options/io-mode

Sets the I/O mode for writing CFF files.

1. `HOST`: I/O is done serially by the host process.

2. `NODE0`: I/O is done serially by the node 0 process.

3. `PARALLEL INDEPENDENT`: I/O is done in parallel using the independent mode of MPI I/O.

4. `PARALLEL COLLECTIVE`: I/O is done in parallel using the collective mode of MPI I/O.

file/cffio-options/single-precision-data?

Specifies whether the double-precision solver saves single-precision data when writing CFF data files, in order to reduce the size of the files.

file/close-without-save?

Exits Ansys Fluent without saving data in Workbench. This command is only available when running Ansys Fluent in Workbench.

file/confirm-overwrite?

Confirms attempts to overwrite existing files.

`file/data-file-options`

Sets derived quantities to be written in data file.

`file/define-macro`

Saves input to a named macro.

`file/em-mapping/`

Enters the electromagnetic loss mapping menu.

> **Important:**
>
> When Ansys Fluent is run under Workbench and a connection is detected between the Ansys Fluent and Ansys Maxwell applications, the `em-mapping` text command menu includes only the `volumetric-energy-loss` and `surface-energy-loss` commands. The `maintain-loss-on-initialization` and `remove-loss-only` commands are available when there is no connection between Ansys Fluent and Ansys Maxwell and Fluent has electromagnetic loss data; Fluent can receive this loss data through a connection or from a previously solved data file.

`file/em-mapping/maintain-loss-on-initialization`

Maintains the loss data provided by Maxwell even if solution is initialized.

`file/em-mapping/remove-loss-only`

Removes the loss data provided by Maxwell and keeps all other solution data.

`file/em-mapping/surface-energy-loss`

Maps the total surface loss (that is, heat source) from 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 Maxwell applications.

`file/em-mapping/volumetric-energy-loss`

Maps the total volumetric loss (that is, heat source) from 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 Maxwell applications.

`file/execute-macro`

Runs a previously defined macro.

`file/export-to-cfd-post`

Exports data files that are compatible with CFD-Post and EnSight (that is, `.cdat` and `.cst` files) and opens CFD-Post, if desired.

`file/export/`

Exports case and data information.

file/export/abaqus

Writes an ABAQUS file.

file/export/ascii

Writes an ASCII file.

file/export/avs

Writes an AVS UCD file.

file/export/cdat-for-cfd-post-&-ensight

Writes data files that are compatible with CFD-Post and EnSight (that is, `.cdat` and `.cst` files).

file/export/cgns

Writes a CGNS file.

file/export/common-fluids-format-post

Writes common fluids format files (`.cas.post` and `.dat.post`) for mesh and data, respectively. These files are saved in the hierarchical data format.

file/export/custom-heat-flux

Writes a generic file for heat transfer.

file/export/dx

Writes an IBM Data Explorer format file.

file/export/ensight

Writes EnSight geometry, velocity, and scalar files.

file/export/ensight-gold

Writes EnSight Gold geometry, velocity, and scalar files.

file/export/ensight-gold-parallel-surfaces

Writes EnSight Gold geometry, velocity, and scalar files for surfaces in a parallel format suitable for EnSight Enterprise.

file/export/ensight-gold-parallel-volume

Writes EnSight Gold geometry, velocity, and scalar files for cell zones in a parallel format suitable for EnSight Enterprise.

file/export/ensight-gold-transient

Writes EnSight Gold Transient geometry, velocity, and scalar files.

`file/export/`fast-mesh

Writes FAST/Plot3D unstructured mesh file.

`file/export/`fast-scalar

Writes FAST/Plot3D unstructured scalar function file.

`file/export/`fast-solution

Writes FAST/Plot3D unstructured solution file.

`file/export/`fast-velocity

Writes FAST/Plot3D unstructured vector function file.

`file/export/`fieldview

Writes FIELDVIEW case and data files.

`file/export/`fieldview-data

Writes FIELDVIEW case and data files.

`file/export/`fieldview-unstruct

Writes FIELDVIEW unstructured combined file.

`file/export/`fieldview-unstruct-mesh

Writes FIELDVIEW unstructured mesh-only file.

`file/export/`fieldview-unstruct-data

Writes FIELDVIEW unstructured results-only file.

`file/export/`fieldview-unstruct-surfaces

Writes FIELDVIEW unstructured file for surfaces. You are prompted to select either **[1]**, **[2]** or **[3]** to write either mesh-only, results-only, or combined for surfaces (respectively).

`file/export/`gambit

Writes GAMBIT neutral file.

`file/export/`icemcfd-for-icepak

Writes a binary ICEM CFD domain file.

`file/export/`ideas

Writes an I-deas universal file.

`file/export/`mechanical-apdl-input

Writes a Mechanical APDL Input file.

`file/export/`nastran

> Writes a NASTRAN file.

`file/export/`particle-history-data

> Exports particle-history data.

`file/export/`patran-neutral

> Writes a PATRAN neutral file.

`file/export/`patran-nodal

> Writes a PATRAN nodal results file.

`file/export/`system-coupling-definition-file-settings/

> Enters the system coupling file (`.scp`) menu.

> **`file/export/system-coupling-definition-file-settings/`enable-automatic-creation-of-scp-file**
>
> > enables/disables automatically writing a `.scp` every time a case file is written.

> **`file/export/system-coupling-definition-file-settings/`write-system-coupling-file**
>
> > Writes a system coupling (`.scp`) file.

`file/export/`taitherm

> Exports TAITherm file.

`file/export/`tecplot

> Writes a Tecplot+3DV format file.

`file/`fsi/

Enters the fluid-structure interaction menu.

`file/fsi/`display-fsi-mesh

> Displays the mesh for a fluid-structure interaction.

`file/fsi/`read-fsi-mesh

> Reads an FEM mesh for one-way data mapping from Ansys Fluent.

`file/fsi/`write-fsi-mesh

> Writes a fluid-structure interaction mesh file.

file/import/

Imports case and data information.

file/import/**abaqus/**

Imports an ABAQUS file.

file/import/abaqus/**fil**

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

file/import/abaqus/**input**

Reads an ABAQUS input file as a case file.

file/import/abaqus/**odb**

Reads an ABAQUS odb file as a case file.

file/import/**cfx/**

Imports a CFX file.

file/import/cfx/**definition**

Reads a CFX definition file as a case file.

file/import/cfx/**result**

Reads a CFX definition file as a case file.

file/import/**cgns/**

Imports a CGNS file.

file/import/cgns/**data**

Reads data from CGNS file.

file/import/cgns/**mesh**

Imports a CGNS mesh file.

file/import/cgns/**mesh-data**

Imports a CGNS mesh file and data file.

file/import/**chemkin-mechanism**

Reads a CHEMKIN mechanism file.

file/import/**chemkin-report-each-line?**

Enables/disables reporting after reading each line.

`file/import/ensight`

Reads an EnSight file as a case file.

`file/import/fidap`

Imports a FIDAP neutral file.

`file/import/flamelet/`

Imports a flamelet file.

`file/import/flamelet/standard`

Reads a standard format flamelet file.

`file/import/flamelet/cfx-rif`

Reads a CFX-RIF format flamelet file.

`file/import/fluent4-case`

Imports a formatted Ansys Fluent 4 case file.

`file/import/fmu-file/`

Enters the import FMU file menu.

`file/import/fmu-file/define-fmu`

Links the FMU variables with the Fluent output parameters.

`file/import/fmu-file/import-fmu`

Imports an FMU file.

`file/import/fmu-file/select-fmu-local`

Allows you to select FMU local variables to be monitored during the calculation.

`file/import/fmu-file/set-fmu-parameter`

Allows you to select FMU parameter variables and change their values.

`file/import/gambit`

Imports a GAMBIT neutral file.

`file/import/hypermesh`

Reads a HYPERMESH file as a case file.

`file/import/ideas-universal`

Imports an I-deas Universal file.

file/import/lstc/

Imports an LSTC file.

file/import/lstc/input

Reads an LSTC input file as a case file.

file/import/lstc/state

Reads an LSTC result file as a case file.

file/import/marc-post

Reads a MARC POST file as a case file.

file/import/mechanical-apdl/

Imports a Mechanical APDL file.

file/import/mechanical-apdl/input

Reads a Mechanical APDL file as a case file.

file/import/mechanical-apdl/result

Reads a Mechanical APDL result file as a case file.

file/import/nastran/

Imports a NASTRAN file.

file/import/nastran/bulkdata

Reads a NASTRAN file as a case file.

file/import/nastran/output2

Reads a NASTRAN op2 file as a case file.

file/import/partition/

Enters the partition menu to set conditions for partitioning an Ansys Fluent case file during read.

file/import/partition/metis

Reads and partitions an Ansys Fluent case file.

file/import/partition/metis-zone

Reads and partitions an Ansys Fluent case file.

file/import/patran/

Imports a PATRAN neutral file (zones defined by named components).

`file/import/patran/neutral`

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

`file/import/plot3d/`

Imports a PLOT3D file.

`file/import/plot3d/mesh`

Reads a PLOT3D file as a case file.

`file/import/tecplot/`

Enters the Tecplot menu.

`file/import/tecplot/mesh`

Reads a Tecplot binary file as a case file.

`file/import/prebfc-structured`

Imports a formatted PreBFC structured mesh file.

`file/import/ptc-mechanica`

Reads a PTC Mechanica Design file as a case file.

`file/load-act-tool`

Loads the Ansys ACT simulation customization tool.

`file/interpolate/`

Interpolates data to/from another grid.

`file/interpolate/read-data`

Reads and interpolates data.

`file/interpolate/write-data`

Writes data for interpolation.

`file/interpolate/zone-selection`

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

`file/read-case`

Reads a case file.

`file/read-case-data`

Reads a case and a data file.

file/read-data

> Reads a data file.

file/read-field-functions

> Reads custom field function definitions from a file.

file/read-injections

> Reads all DPM injections from a file.

file/read-isat-table

> Reads ISAT Table.

file/read-journal

> Reads command input from one or more files.

file/read-macros

> Reads macro definitions from a file.

file/read-pdf

> Reads a PDF file.

file/read-profile

> Reads boundary profile data.

file/read-rays

> Reads a ray file.

file/read-settings

> Reads and sets boundary conditions from a specified file.

file/read-surface-clusters

> Reads surface clusters from a file.

file/read-transient-table

> Reads table of transient boundary profile data.

file/read-viewfactors

> Reads view factors from a file.

file/reload-setup

Discards any changes 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.

file/replace-mesh

Replaces the mesh with a new one while preserving settings.

file/set-batch-options

Sets the batch options.

file/set-idle-timeout

Allows you to set an idle timeout so that an idle Ansys Fluent session will automatically save and close after the specified time.

file/set-tui-version

Allows you to improve backwards compatibility for journal files. This command hides any new TUI prompts that are added at a future release of Ansys Fluent and reverts to the arguments of the release that you specify using the command (within two full releases of the current release). The command is automatically added to a journal file as soon as you start the recording.

file/show-configuration

Displays current release and version information.

file/single-precision-coordinates?

Specifies whether the nodal coordinates should be written in single precision (rather than the default double precision). This text command is only available in the single-precision version of Ansys Fluent.

file/solution-files/

Enters the solution files menu.

file/solution-files/delete-solution

Deletes solution files.

file/solution-files/load-solution

Loads a solution file.

file/solution-files/print-solution-files

Prints a list of available solution files.

file/start-journal

Starts recording all input in a file. Note that commands entered using paths from older versions of Fluent will be upgraded to their current path in the journal file.

file/start-transcript

Starts recording input and output in a file.

file/stop-journal

Stops recording input and closes journal file.

file/stop-macro

Stops recording input to a macro.

file/stop-transcript

Stops recording input and output and closes transcript file.

file/sync-workbench

Directly updates Workbench with the most recent Fluent changes. This command is only available when running Ansys Fluent in Workbench.

file/table-manager

Enters the table file manager menu.

file/table-manager/delete

Deletes a table from local storage.

file/table-manager/list-matrix-data

Lists the data of a "matrix" type locally-stored table.

file/table-manager/list-properties

Lists the properties of a locally-stored table.

file/table-manager/list-tables

Lists all locally-stored tables.

file/table-manager/read-matrix-data-file

Imports a "matrix" type table from a file into local storage.

file/table-manager/read-rgp-file

Imports a set of Real Gas Property (RGP) tables from a file into local storage.

file/table-manager/rename

Renames a locally-stored table.

file/table-manager/store-in-case-file?

Sets the persistence mode for RGP tables (embedded in case file or rely on external file).

file/transient-export/

file/transient-export/abaqus

Writes an ABAQUS file.

file/transient-export/ascii

Writes an ASCII file.

file/transient-export/avs

Writes an AVS UCD file.

file/transient-export/cdat-for-cfd-post-&-ensight

Writes data files that are compatible with CFD-Post and EnSight (that is, `.cdat` and `.cst` files).

file/transient-export/cgns

Writes a CGNS file.

file/transient-export/dx

Writes an IBM Data Explorer format file.

file/transient-export/ensight-gold-transient

Writes EnSight Gold geometry, velocity, and scalar files.

file/transient-export/ensight-gold-from-existing-files

Writes EnSight Gold files using Ansys Fluent case files.

file/transient-export/ensight-gold-parallel-surfaces

Writes EnSight Gold geometry, velocity, and scalar files for surfaces in a parallel format suitable for EnSight Enterprise.

file/transient-export/ensight-gold-parallel-volume

Writes EnSight Gold geometry, velocity, and scalar files for cell zones in a parallel format suitable for EnSight Enterprise.

file/transient-export/fast

Writes a FAST/Plot3D unstructured mesh velocity scalar file.

file/transient-export/fast-solution

Writes a FAST/Plot3D unstructured solution file.

file/transient-export/fieldview-unstruct

Writes a FIELDVIEW unstructured combined file.

file/transient-export/fieldview-unstruct-mesh

Writes a FIELDVIEW unstructured mesh only file.

file/transient-export/fieldview-unstruct-data

Writes a FIELDVIEW unstructured results only file.

file/transient-export/fieldview-unstruct-surfaces

Writes FIELDVIEW unstructured combined file for surfaces.

file/transient-export/ideas

Writes an I-deas universal file.

file/transient-export/mechanical-apdl-input

Writes a Mechanical APDL input file.

file/transient-export/nastran

Writes a NASTRAN file.

file/transient-export/patran-neutral

Writes a PATRAN neutral file.

file/transient-export/taitherm

Writes a TAITherm file.

file/transient-export/particle-history-data

Sets up an automatic particle-history data export.

file/transient-export/edit

Edits transient exports.

file/transient-export/delete

Deletes transient exports.

file/transient-export/settings/

Enters the automatic export settings menu.

file/transient-export/settings/cfd-post-compatible

Specifies 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.

`file/`write-boundary-mesh

Writes the boundary mesh to a file.

`file/`write-case

Writes a case file.

`file/`write-case-data

Writes a case and a data file.

`file/`write-cleanup-script

Writes the cleanup-script-file for Ansys Fluent.

`file/`write-data

Writes a data file.

`file/`write-fan-profile

Computes radial profiles for a fan zone and writes them to a profile file.

`file/`write-field-functions

Writes the currently defined custom field functions to a file.

`file/`write-flamelet

Writes a flamelet file.

`file/`write-injections

Writes out selected DPM injections to a file.

`file/`write-isat-table

Writes ISAT Table.

`file/`write-macros

Writes the currently defined macros to a file.

`file/`write-merge-profiles

Writes a .csv file with the selected surfaces consolidated into one set of data points.

`file/`write-pdat?

Enables / disables the attempt to save `.pdat` files. Note that this text command is no longer supported.

`file/`write-pdf

Writes a pdf file.

`file/`write-profile

Writes surface data as a boundary profile file.

`file/`write-settings

Writes out current boundary conditions in use.

429

Chapter 6: `mesh/`

`mesh/adapt/`

Enters the mesh adaption menu.

`mesh/adapt/`**adapt-mesh**

Adapts the mesh according to the methods and settings that you specified.

`mesh/adapt/`**anisotropic-adaption**

Applies legacy anisotropic adaption to refine the boundary layers or registers. Cells will be split in the normal direction to the boundary face. Note that this text command is only available for 3D cases that have the adaption method set to hanging node.

`mesh/adapt/`**cell-registers/**

Enters the cell registers menu.

`mesh/adapt/cell-registers/`**add**

Creates a new cell register.

`mesh/adapt/cell-registers/`**apply-poor-mesh-numerics**

Applies poor mesh numerics to the mesh of a cell register.

`mesh/adapt/cell-registers/`**coarsen**

Coarsen the mesh based on a cell register.

`mesh/adapt/cell-registers/`**delete**

Deletes a cell register.

`mesh/adapt/cell-registers/`**display**

Displays a cell register.

`mesh/adapt/cell-registers/`**edit**

Edits an existing cell register.

`mesh/adapt/cell-registers/`**list**

Lists all of the currently defined cell registers.

`mesh/adapt/cell-registers/`list-properties

Lists the properties of a cell register.

`mesh/adapt/cell-registers/`refine

Refine the mesh based on a cell register.

`mesh/adapt/`coarsening-criteria

Allows you to provide an expression for the coarsening criterion.

`mesh/adapt/`display-adaption-cells

Displays the cells that are marked for adaption in the graphics window.

`mesh/adapt/`free-hierarchy

Deletes the defined adaption hierarchy.

`mesh/adapt/`geometry/

Enters the geometry menu. Note that this text command menu is not available unless the adaption method is set to hanging node.

`mesh/adapt/geometry/`reconstruct-geometry

Enables/disables geometry-based adaption.

`mesh/adapt/geometry/`set-geometry-controls

Sets geometry controls for wall zones.

`mesh/adapt/`list-adaption-cells

Prints the number of cells marked for refinement, coarsening, and both to the console.

`mesh/adapt/`manage-criteria/

Enters the manage criteria menu, which provides text commands for managing automatic adaption criteria.

`mesh/adapt/manage-criteria/`add

Adds a new automatic adaption criterion.

`mesh/adapt/manage-criteria/`delete

Deletes an existing automatic adaption criterion.

`mesh/adapt/manage-criteria/`edit

Edits an existing automatic adaption criterion.

mesh/adapt/manage-criteria/list

Lists all the existing automatic adaption criteria.

mesh/adapt/manage-criteria/list-properties

Lists the properties of an existing automatic adaption criterion.

mesh/adapt/predefined-criteria/

Enters the predefined-criteria menu, which allows you to select commonly used criteria for adapting the mesh.

mesh/adapt/predefined-criteria/aerodynamics/

Enters the aerodynamics menu, which provides text commands that create cell registers and define adaption criteria that can be useful for aerodynamic simulations.

mesh/adapt/predefined-criteria/aerodynamics/shock-indicator/

Enters the shock indicator menu, which provides text commands that create cell registers and define adaption criteria that can be useful for simulations with shocks.

mesh/adapt/predefined-criteria/aerodynamics/shock-indicator/density-based

Creates cell registers and defines an adaption criterion that is suitable for simulations with shocks that use the density-based solver or the pressure-based solver with a fluid that uses a real-gas or ideal-gas model for the density.

mesh/adapt/predefined-criteria/combustion/

Enters the combustion menu, which provides text commands that create named expressions and cell registers and define adaption criteria that can be useful for combustion simulations.

mesh/adapt/predefined-criteria/combustion/flame-indicator

Creates named expressions and cell registers and defines adaption criteria that are suitable for combustion simulations, so that the mesh is refined along a progressing flame front using various criteria like temperature, vorticity, species, and DPM concentration (depending on which models are used). There is also an option for refining a spherical spark region prior to a transient run, which after a specified time is then coarsened back to the original mesh.

mesh/adapt/predefined-criteria/multiphase/

Enters the multiphase menu, which provides text commands that create named expressions and cell registers and define adaption criteria that can be useful for Volume of Fluid (VOF) simulations.

mesh/adapt/predefined-criteria/multiphase/vof

Creates a named expression and cell registers, and defines adaption settings that are suitable for standard Volume of Fluid (VOF) simulations.

mesh/adapt/predefined-criteria/multiphase/vof-to-dpm-advanced

Sets up adaption that is suitable when using the VOF-to-DPM model transition mechanism; the resulting adaption criteria will be defined by complex expressions that draw upon cell registers, as well as parameters that you will need to define.

mesh/adapt/predefined-criteria/multiphase/vof-to-dpm-generic

Sets up adaption that is suitable when using the VOF-to-DPM model transition mechanism; the resulting adaption criteria will be defined by a named expression and cell registers that are fairly straightforward, as well as parameters that you will need to define.

mesh/adapt/profile/

Enters the profile menu.

mesh/adapt/profile/clear

Clears the adaption profiling counters.

mesh/adapt/profile/disable

Disables adaption profiling.

mesh/adapt/profile/enable

Enables adaption profiling.

mesh/adapt/profile/print

Prints adaption profiling results.

mesh/adapt/refinement-criteria

Allows you to provide an expression for the refinement criterion.

mesh/adapt/set/

Enters the set menu.

mesh/adapt/set/additional-refinement-layers

Allows you to specify additional refinement layers (this is an advanced control).

mesh/adapt/set/anisotropic-adaption?

Enables / disables anisotropic adaption. Note that this text command requires that the adaption method is set to PUMA.

mesh/adapt/set/anisotropic-boundary-zones

Allows you to select the boundary zones that specify directions for anisotropic refinement with the PUMA method.

`mesh/adapt/set/`anisotropic-split-ratio

Sets the split ratio for the cells as part of anisotropic refinement with the PUMA method.

`mesh/adapt/set/`cell-zones

Sets cell zones to be used for marking adaption. An empty list specifies that all zones are considered for adaption.

`mesh/adapt/set/`display-settings

Sets the graphics display options for the refinement, coarsening, and common cells.

`mesh/adapt/set/`maximum-cell-count

Sets an approximate limit to the total cell count of the mesh during adaption. Fluent uses this value to determine when to stop marking cells for refinement. A value of zero places no limits on the number of cells.

`mesh/adapt/set/`maximum-refinement-level

Controls the number of levels of refinement used to split cells during the adaption.

`mesh/adapt/set/`method

Sets the adaption method.

`mesh/adapt/set/`minimum-cell-quality

Sets the minimum value allowed for the orthogonal quality of cells during adaption. If your solution diverges, you may find that using a higher minimum quality value resolves the issue. This text command is only available with the PUMA 3D adaption method.

`mesh/adapt/set/`minimum-edge-length

sets an approximate limit to the edge length for cells that are considered for refinement. Even if a cell is marked for refinement, it will not be refined if (for 3D) its volume is less than the cube of this field or (for 2D) its area is less than the square of this field. The default value of zero places no limits on the size of cells that are refined.

`mesh/adapt/set/`overset-adapt-dead-cells?

Enables/disables the adaption of dead cells in overset meshes.

`mesh/adapt/set/`verbosity

Allows you set how much information about the adaption is printed to the console.

`mesh/`adjacency

Views and renames face zones adjacent to selected cell zones.

`mesh/`check

Performs various mesh consistency checks and displays 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`).

`mesh/check-before-solve`

The default value for `mesh/check-before-solve` is "no". If `mesh/check-before-solve` is set to "yes", a mesh check operation will be invoked prior to starting solver. If grid check fails, solver will be interrupted, and relevant information will be printed in the Fluent console.

`mesh/`check-verbosity

Sets 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 enables the availability of additional mesh field variables, 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/`memory-usage

Reports solver memory use.

`mesh/`mesh-info

Prints zone information size.

`mesh/`modify-zones/

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

`mesh/`polyhedra/

Enters the polyhedra menu.

`mesh/polyhedra/`convert-domain

Converts the entire domain to polyhedra cells.

`mesh/polyhedra/`convert-hanging-nodes

Converts cells with hanging nodes/edges to polyhedra.

`mesh/polyhedra/`convert-skewed-cells

Converts skewed cells to polyhedra.

`mesh/polyhedra/`**`options/`**

Enters the polyhedra options menu.

`mesh/polyhedra/options/`**`migrate-and-reorder?`**

Enables / disables the migration of newly created partitions to the compute-nodes and the reordering of the domain as part of polyhedra conversion. This is disabled by default, because it requires significant additional memory; when disabled, it is recommended that you save the case file after conversion, read it in a new Fluent session (so that the new / stored partitions become active), and then manually reorder using the `mesh/reorder/reorder-domain` text command. If you want to run the calculation in the current Fluent session you can enable the `migrate-and-reorder?` text command prior to conversion, but you must ensure that no more than half of the available memory of your system is currently used.

`mesh/polyhedra/options/`**`preserve-boundary-layer?`**

Specifies whether boundary layer cells will be preserved when the domain is converted to polyhedra. When the value is set to `0` (default) Ansys Fluent checks for high aspect ratio cells at the boundary layer and if any are found, Fluent asks if you want to preserve the boundary layer. When the value is set to `1`, the boundary layer cells are never preserved; when it is set to `2`, the boundary layer cells are always preserved (regardless of the aspect ratio of the boundary layer cells).

`mesh/polyhedra/options/`**`preserve-interior-zones`**

Enables 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.

`mesh/`**`quality`**

Displays 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`.

`mesh/`**`redistribute-boundary-layer`**

Redistributes the nodes in a boundary layer zone to achieve a desired growth rate after anisotropic adaption.

`mesh/`**`reorder/`**

Reorders domain menu.

`mesh/reorder/`**`band-width`**

Prints cell bandwidth.

`mesh/reorder/`reorder-domain

Reorders cells and faces using the reverse Cuthill-McKee algorithm. Note that you must save a new case file (and a data file, if data exists) after reordering with this text command, as well as recreate any ray files and/or surface cluster information.

`mesh/reorder/`reorder-zones

Reorders zones by partition, type, and ID.

`mesh/`repair-improve/

`mesh/repair-improve/`allow-repair-at-boundaries

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

`mesh/repair-improve/`improve-quality

Improves poor quality cells in the mesh, if possible.

`mesh/repair-improve/`include-local-polyhedra-conversion-in-repair

Enables/disables 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.

`mesh/repair-improve/`repair

Repairs 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.

`mesh/repair-improve/`repair-face-handedness

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

`mesh/repair-improve/`repair-face-node-order

Modifies face nodes to repair faces with improper face node order and, therefore, eliminates any resulting left-handed faces.

`mesh/repair-improve/`repair-periodic

Modifies 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.

`mesh/repair-improve/repair-wall-distance`

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

`mesh/repair-improve/report-poor-elements`

Reports invalid and poor quality elements.

`mesh/rotate`

Rotates the mesh.

`mesh/scale`

Prompts for the scaling factors in each of the active Cartesian coordinate directions.

`mesh/size-info`

Prints mesh size.

`mesh/smooth-mesh`

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

`mesh/surface-mesh/`

Enters the Surface Mesh menu.

`mesh/surface-mesh/delete`

Deletes surface mesh.

`mesh/surface-mesh/display`

Displays surface meshes.

`mesh/surface-mesh/read`

Reads surface meshes.

`mesh/swap-mesh-faces`

Swaps mesh faces.

`mesh/translate`

Prompts for the translation offset in each of the active Cartesian coordinate directions.

Chapter 7: `parallel/`

`parallel/bandwidth`

Shows network bandwidth.

`parallel/check`

Performs checks of various factors that affect parallel performance.

`parallel/check`

Sets verbosity output of the parallel check. Higher verbosity corresponds to more detailed information.

`parallel/gpgpu/`

Enters the GPGPU menu

> **`parallel/gpgpu/select`**
>
> Selects which GPGPUs to use for AMG acceleration
>
> **`parallel/gpgpu/show`**
>
> Lists the available GPGPUs. GPGPUs selected for use are indicated by the presence of an asterisk (*).

`parallel/latency`

Shows network latency.

`parallel/load-balance/`

Enters the load balancing parameters menu.

> **`parallel/load-balance/physical-models`**
>
> Uses physical-models load balancing?
>
> **`parallel/load-balance/dynamic-mesh`**
>
> Uses load balancing for dynamic mesh?
>
> **`parallel/load-balance/mesh-adaption`**
>
> Uses load balancing for mesh adaption?

`parallel/`multidomain/

Enters the multidomain architecture menu.

`parallel/multidomain/`conjugate-heat-transfer/

Enters the conjugate heat transfer menu.

`parallel/multidomain/conjugate-heat-transfer/`enable?

Enables/disables loosely coupled conjugate heat transfer. This works in conjunction with the `solve/set/transient-controls/solid-time-step-size` text command (and thereby enables the use of a user-specified time step size for solid zones that can be larger than that used for the fluid zones) to increase the robustness of the energy equation calculation, and specifies that multidomain architecture is used within a single Fluent session to enhance the performance of the simulation.

`parallel/multidomain/conjugate-heat-transfer/`set/

Enters the set menu for loosely coupled conjugate heat transfer.

`parallel/multidomain/conjugate-heat-transfer/set/`coupling

Specifies when the fluid and solid zone calculations are coupled, either at a defined time period or number of fluid time steps.

`parallel/`partition/

Enters the partition domain menu.

`parallel/partition/`auto/

Sets auto partition parameters.

`parallel/partition/auto/`across-zones

Enables auto partitioning by zone or by domain.

`parallel/partition/auto/`load-vector

Sets the auto partition load vector.

`parallel/partition/auto/`method

Sets the partition method.

`parallel/partition/auto/`pre-test

Sets auto partition pre-testing optimization.

`parallel/partition/auto/`use-case-file-method

Uses partitions in a pre-partitioned case file.

parallel/partition/combine-partition

Merges every N partitions.

parallel/partition/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.)

parallel/partition/method

Sets the partition method.

parallel/partition/print-active-partitions

Prints active partition information (parallel solver).

parallel/partition/print-stored-partitions

Prints stored partition information (parallel solver).

parallel/partition/reorder-partitions

Reorders partitions.

parallel/partition/reorder-partitions-to-architecture

Reorders partitions to architecture.

parallel/partition/set/

Enters the set partition parameters menu.

parallel/partition/set/across-zones

Allows 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.

parallel/partition/set/all-off

Disables all optimizations.

parallel/partition/set/all-on

Enables all optimizations.

parallel/partition/set/cell-function

Sets cell function.

parallel/partition/set/dpm-load-balancing

Enables / disables dynamic load balancing for discrete phase model cases that use a second domain for DPM particle tracking (that is, cases for which you have enabled the `define/models/dpm/parallel/hybrid-2domain?` text command).

parallel/partition/set/face-area-as-weights

Uses face area as connection weights.

parallel/partition/set/isat-weight

Sets ISAT weight.

laplace-smoothing

Enables the Laplace smoothing option which can prevent partition boundaries from lying along regions of high aspect ratio cells.

parallel/partition/set/load-distribution

Sets 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.

parallel/partition/set/merge

Toggles the optimizer that attempts to decrease the number of interfaces by eliminating orphan cell clusters.

parallel/partition/set/model-weighted-partition

Enables / disables model-weighted partitioning. This option works with the METIS partitioning method, and specifies that Fluent automatically calculates the weighting based on the cell count and the models and attributes specified as weights (using the `parallel/partition/set/isat-weight` text command, for example).

parallel/partition/set/nfaces-as-weights

Uses number of faces as weights.

parallel/partition/set/origin

Sets 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).

parallel/partition/set/particle-weight

Sets DPM particle weight.

parallel/partition/set/pre-test

Enables the operation that determines the best coordinate-splitting direction.

parallel/partition/set/solid-thread-weight

Uses solid thread weights.

parallel/partition/set/smooth

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

parallel/partition/set/verbosity

Controls 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.

parallel/partition/set/vof-free-surface-weight

Sets VOF free surface weight.

parallel/partition/smooth-partition

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

parallel/partition/use-stored-partitions

Uses this partitioning.

parallel/set/

Enters the set parallel parameters menu.

parallel/set/fast-i/o?

Uses the fast I/O option.

parallel/set/partition-mask

Sets partition mask.

parallel/set/time-out

Sets spawn time-out in seconds.

parallel/set/verbosity

Sets the parallel verbosity.

parallel/show-connectivity

Prints the network connectivity for the selected compute node.

parallel/thread-number-control

Sets the maximum number of threads on each machine.

parallel/timer/

Enters the timer menu.

parallel/timer/usage

Prints performance statistics in the console window.

parallel/timer/reset

Adjusts domain timers.

Chapter 8: `plot/`

`plot/change-fft-ref-pressure`

Changes reference acoustic pressure.

`plot/circum-avg-axial`

Computes iso-axial band surfaces and plots data vs. axial coordinate on them.

`plot/circum-avg-radial`

Computes iso-radial band surfaces and plots data vs. radius on them.

`plot/cumulative-plot/`

Plot the development of force, force coefficient, moment, or moment coefficient across the specified wall zones.

`plot/cumulative-plot/add`

Create a new cumulative plot.

`plot/cumulative-plot/delete`

Delete an existing cumulative plot object.

`plot/cumulative-plot/edit`

Edit an existing cumulative plot object.

`plot/cumulative-plot/list`

Print the names of the existing cumulative plot objects to the console.

`plot/cumulative-plot/list-properties`

Print the properties of the specified cumulative plot object to the console.

`plot/cumulative-plot/plot`

Plot a cumulative plot in the graphics window.

`plot/cumulative-plot/print`

Print the value of a cumulative plot to the console.

plot/cumulative-plot/write

Write a cumulative plot to a file.

plot/datasources/

Enters the menu for creating and modifying plots containing multiple data sources.

plot/datasources/add-report-plot

Specify an existing report plot to include in this plot.

plot/datasources/add-xy-plot

Specify an existing XY plot to include in this plot.

plot/datasources/list-curve-information

Prints the names of the variables selected for plotting.

plot/datasources/list-data-sources

Prints the names of the selected/loaded data sources including the files and in-session plot objects.

plot/datasources/list-legend-names

Prints the names of the legends to the console.

plot/datasources/load-file

Specify a file to load for plotting. The file must be located in the working directory.

plot/datasources/plot

Plots the selected data in the active graphics window.

plot/datasources/properties

Prints the properties of whichever plot or file included in the multiple data sources plot that you specify.

plot/datasources/remove-data-source

Removes whichever data source (plot or file) you specify.

plot/datasources/rename-legend

Allows you to rename legend entries. Note that you must re-plot to see the legend name change.

plot/display-profile-data

Plots profile data.

plot/fft

Plots FFT of file data.

plot/file

Plots data from an external file.

plot/file-list

Plots data from multiple external files.

plot/file-set/

Sets file plot parameters.

plot/file-set/auto-scale?

Sets 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.

plot/file-set/background-color

Sets the color of the field within the abscissa and ordinate axes.

plot/file-set/key

Enables/disables display of curve key and sets its window title.

plot/file-set/file-lines

Sets parameters for plot lines.

plot/file-set/file-markers

Sets parameters for data markers.

plot/file-set/labels

Sets labels for plot axes.

plot/file-set/lines

Sets parameters for plot lines.

plot/file-set/log?

Uses log scales for one or both axes.

plot/file-set/markers

Sets parameters for data markers.

plot/file-set/numbers

Sets number formats for axes.

`plot/file-set/`plot-to-file

Specifies a file in which to write XY plot data.

`plot/file-set/`rules

Sets parameters for display of major and minor rules.

`plot/file-set/`windows/

XY plot window options. For a description of the items in this menu, see `display/set/win-dows/xy`.

`plot/`flamelet-curves/

Enters the flamelet curves menu.

`plot/flamelet-curves/`plot-curves

Plots of a curve property.

`plot/flamelet-curves/`write-to-file?

Writes curve to a file instead of plot.

`plot/`histogram

Plots a histogram of the specified solution variable using the defined range and number of intervals.

`plot/`histogram-set/

Sets histogram plot parameters. Sub-menu items are the same as `file-set/` above.

`plot/`label-alignment

Set the orientation of XY plot axis labels as either horizontal or axis-aligned.

`plot/`plot

Plots solution on surfaces.

`plot/`plot-direction

Sets plot direction for XY plot.

`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.

`plot/`residuals-set/

Sets residual plot parameters. Sub-menu items are the same as `file-set/` above.

`plot`/set-boundary-val-off

Disables the use of boundary face values when node values are disabled in solution XY plots. This option is disabled by default, that is, boundary face values are used when node values are disabled.

`plot`/solution

Plots solution on surfaces and/or zones. Zone and surface names can be indicated using a wildcard (*).

`plot`/solution-set/

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

`plot`/vrx-sound-analysis/

Enter the VRXpereince sound analysis menu.

`plot/vrx-sound-analysis/`print-indicators

Read in a pressure signal or spectrum file, then optionally read in a frequency response function (FRF) or transfer function file, before printing the acoustics indicators.

`plot/vrx-sound-analysis/`write-files

Read in a pressure signal or spectrum file, then optionally read in a frequency response function (FRF) or transfer function file, before writing a WAV, output pressure, and/or acoustic indicators file.

Chapter 9: `preferences/`

Important:

Values that correspond to drop-down entries in the GUI version of **Preferences** are case-sensitive must be entered with the same capitalization as the GUI version and be contained within quotation marks.

`preferences/`appearance/

Enter the menu for preferences covering appearance.

> **`preferences/appearance/`ansys-logo/**
>
> Enter the menu for controlling Ansys logo visibility.
>
> > **`preferences/appearance/ansys-logo/`visible**
> >
> > Enable or disable the visibility of the Ansys logo in the graphics window.
> >
> > **`preferences/appearance/ansys-logo/`color**
> >
> > Specify whether the Ansys logo is white or black.
>
> **`preferences/appearance/`axis-triad**
>
> Enable or disable the visibility of the axis triad in the graphics window.
>
> **`preferences/appearance/`charts/**
>
> Enter the menu for controlling the display of 2D charts/plots.
>
> > **`preferences/appearance/charts/`curve-colors**
> >
> > Specify the initial set of default colors for the rendering of curves. *Note that changing this setting requires any plots to be replotted before you see the effect of the new setting.*
> >
> > **`preferences/appearance/charts/`modern-plots-enabled**
> >
> > Enables enhanced plots, which is a beta feature. Enabling this feature exposes new fields (all beta functionality).
>
> **`preferences/appearance/`color-theme**
>
> Specify a color theme for the appearance of Ansys Fluent.

---

preferences/appearance/completer

Enable/disable the console automatic-completer, which suggests available commands as you type in the console.

preferences/appearance/graphics-background-color1

Controls the primary background color of the graphics window.

preferences/appearance/graphics-background-color2

Controls the secondary background color when the style is set as a gradient.

preferences/appearance/graphics-background-style

Specify whether the background color is uniform or if there is a gradient.

preferences/appearance/graphics-background-theme

Provides predefined color and style selections for the graphics window background color.

preferences/appearance/graphics-foreground-color

Specify the color of graphics window text.

preferences/appearance/graphics-view

Specify whether the default view is orthographic or perspective.

preferences/appearance/graphics-wall-face-color

Set the default face color for when the mesh is displayed.

preferences/appearance/group-by-tree-view

Specify how boundary conditions are grouped in the tree.

preferences/appearance/number-of-files-recently-used

Controls how many recently-used files are listed in the **File** ribbon tab and the Fluent Launcher.

preferences/appearance/quick-property-view

Enables/Disables the "quick-edit" properties panels that appear when you select a boundary in the graphics windows.

preferences/appearance/ruler

Adds or removes the ruler from the graphics window. *Note that you must be in **orthographic** view for the ruler to be visible in the graphics window.*

preferences/appearance/selections/

Enters the menu for controlling selections in the graphics window.

preferences/appearance/selections/highlight-edge-color

Specifies the color used to highlight edges when the **Hover-Over Highlight** feature is enabled (`mouse-over-highlight-enabled`).

preferences/appearance/selections/highlight-edge-weight

Specifies the thickness of the edge highlights when the **Hover-Over Highlight** feature is enabled (`mouse-over-highlight-enabled`).

preferences/appearance/selections/highlight-face-color

Specify which color indicates that a face is selected.

preferences/appearance/selections/highlight-transparency

Specify the transparency of the coloring on a selected surface. 0.1 is fully opaque and 1 is fully transparent.

preferences/appearance/selections/mouse-hover-probe-enabled

Enable/disable the display of result values in the graphics window upon mouse-over, when graphics objects displaying solution results are displayed.

preferences/appearance/selections/mouse-over-highlight-enabled

Enable/disable the highlighted outline of a surface when hovered-over. Note that objects must be redisplayed after changing this setting before the new setting is visible.

preferences/appearance/show-model-edges

Enable/disable whether mesh edges are shown in a mesh display.

preferences/appearance/startup-page

Enable/disable the display of the startup page when Ansys Fluent is started without loading a mesh or case file.

preferences/appearance/surface-specularity

Specify the specularity of all surfaces except those included in contour plots. Sepecularity is the reflectiveness of a surface; higher values (closer to 1) equate to a more reflective surface.

preferences/appearance/surface-specularity

Specify the specularity of contour surfaces. Sepecularity is the reflectiveness of a surface; higher values (closer to 1) equate to a more reflective surface.

preferences/appearance/titles

Enable/disable the display of solver information in the graphics window.

preferences/general/

Enter the menu for general preferences.

`preferences/general/automatic-transcript`

Enable/disable the automatic creation of a transcript file for each Ansys Fluent session.

`preferences/general/default-ioformat`

Specify the default file format for saving mesh, case, and data files.

`preferences/general/idle-timeout`

Specify the default file format for saving case and data files.

`preferences/general/warn-default-iochange`

Enable/disable the warning message that appears when using case and data files saved in the legacy format.

`preferences/`**graphics/**

Enter the menu for preferences covering appearance.

`preferences/graphics/animation-option`

Specify whether the entire model or just a wireframe is shown during manipulations in the graphics window.

`preferences/graphics/boundary-markers/`

Enter the boundary markers menu.

`preferences/graphics/boundary-markers/color-option`

Specify whether boundary markers are a fixed color or if they match the color of the surface they are identifying.

`preferences/graphics/boundary-markers/enabled`

Enable/disable boundary marker display.

`preferences/graphics/boundary-markers/inlet-color`

Specify the color of the inlet boundary markers.

`preferences/graphics/boundary-markers/marker-fraction`

Specify marker density factor (0.1-1).

`preferences/graphics/boundary-markers/marker-limit`

Specify the maximum number of markers at each boundary.

`preferences/graphics/boundary-markers/outlet-color`

Specify the color of the outlet boundary markers.

`preferences/graphics/boundary-markers/`scale-marker

Specify the scale factor for the boundary markers (0.1-10), which controls the overall size of the markers.

`preferences/graphics/boundary-markers/`show-inlet-markers

Enable/disable the display of boundary markers for inlets.

`preferences/graphics/boundary-markers/`show-outlet-markers

Enable/disable the display of boundary markers for outlets.

`preferences/graphics/`colormap-settings/

Enter the colormap settings menu.

`preferences/graphics/colormap-settings/`alignment

Specify the default colormap location.

`preferences/graphics/colormap-settings/`aspect-ratio-when-vertical

Specify the length vs. width ratio for a vertical colormap, which controls the thickness of the colormap; smaller values mean a thicker colormap.

`preferences/graphics/colormap-settings/`border-style

Specify how/when the colormap border appears.

`preferences/graphics/colormap-settings/`colormap

Choose the default colormap.

`preferences/graphics/colormap-settings/`labels

Specify whether there is a label for every colormap value or if some are skipped.

`preferences/graphics/colormap-settings/`levels

Specify the default colormap size.

`preferences/graphics/colormap-settings/`log-scale

Enable/disable the use of a logarithmic scale for the colormap.

`preferences/graphics/colormap-settings/`major-length-to-screen-ratio-when-vertical

Choose the length of the colormap as a fraction of graphics window height, when the colormap is vertical.

`preferences/graphics/colormap-settings/`number-format-precision

Specify the colormap number label precision.

preferences/graphics/colormap-settings/number-format-type

Specify how colormap numbers are displayed.

preferences/graphics/colormap-settings/show-colormap

Enable/disable the display of colormaps.

preferences/graphics/colormap-settings/skip-value

Specify how many number labels are skipped in the colormap.

preferences/graphics/colormap-settings/text-behavior

Specify whether colormap label text automatically scales with the colormap size.

preferences/graphics/colormap-settings/text-font-automatic-vertical-size

Specify the initial font size as a ratio of the colormap overall size, for vertically aligned colormaps.

preferences/graphics/colormap-settings/text-font-fixed-size

Set the font size for colormap labels.

preferences/graphics/colormap-settings/text-font-name

Set the font for colormap labels.

preferences/graphics/colormap-settings/type

Specify whether the colormap appearance is smooth or banded.

preferences/graphics/double-buffering

Enable/disable double-buffering, which reduces screen flicker, but may use more memory on some machines.

preferences/graphics/graphics-effects/

Enter the graphics effects menu.

preferences/graphics/graphics-effects/anti-aliasing

Enable/disable the smoothing of lines and text.

preferences/graphics/graphics-effects/grid-color

Specify the color of the grid lines when the ground plane grid is shown.

preferences/graphics/graphics-effects/grid-plane-enabled

Enable/disable the display of the ground plane grid.

preferences/graphics/graphics-effects/grid-plane-offset

Set the grid plane offset from the model as a percentage of the model size.

preferences/graphics/graphics-effects/plane-direction

Specify the direction of the plane for the ground plane grid and reflections.

preferences/graphics/graphics-effects/reflections-enabled

Enable/disable model reflections (mirror-type reflections).

preferences/graphics/graphics-effects/shadow-map-enabled

Enable/disable dynamic shadows, which show shadows of geometric entities on other objects based on lighting and object orientation.

preferences/graphics/graphics-effects/show-edge-reflections

Enable/disable the display of model edges in reflections. Note that this can negatively affect performance.

preferences/graphics/graphics-effects/simple-shadows-enabled

Enable/disable the display of static shadows on the ground plane.

preferences/graphics/graphics-effects/update-after-mouse-release

Enable/disable the updating of graphics effects as a model is being manipulated in the graphics window.

preferences/graphics/graphics-window-line-width

Specify the thickness of lines that appear in the graphics window.

preferences/graphics/graphics-window-point-symbol

Specify the symbol used for indicating points in the graphics window (like the points in an XY plot).

preferences/graphics/hardcopy-settings/

Enter the menu for saving picture settings.

preferences/graphics/hardcopy-settings/hardcopy-driver

Specify the default format for saving pictures.

preferences/graphics/hardcopy-settings/hardcopy-line-width

Specify the thinkness of lines for saved pictures.

preferences/graphics/hardcopy-settings/hardcopy-image-accel

Enable/disable the use of hardware acceleration when saving images, where possible.

preferences/graphics/hidden-surface-removal-method

Specify the method for removing hidden surfaces. These methods vary in speed and quality, depending on your machine.

preferences/graphics/lighting/

Enter the lighting menu.

preferences/graphics/lighting/headlight

Turn the headlight on or off or set it as automatic.

preferences/graphics/lighting/headlight-intensity

Specify the intensity of the headlight.

preferences/graphics/lighting/lighting-method

Specify the default lighting method.

preferences/graphics/meshing-mode/

Enter the menu for meshing-specific graphics settings.

preferences/graphics/meshing-mode/graphics-window-display-timeout

Enable/disable graphics window display timeout.

preferences/graphics/meshing-mode/graphics-window-display-timeout-value

Specify the graphics window display timeout value.

preferences/graphics/performance/

Enter the menu for selecting the predefined graphics effects settings.

preferences/graphics/performance/fast-display-mode/

Enter the menu for controlling fast graphics performance preferences.

preferences/graphics/performance/fast-display-mode/faces-shown

Specify whether or not faces are shown when interacting with the model.

preferences/graphics/performance/fast-display-mode/nodes-shown

Specify whether or not nodes are shown when interacting with the model.

preferences/graphics/performance/fast-display-mode/perimeter-edges-shown

Specify whether or not the perimeter edges are shown when interacting with the model.

preferences/graphics/performance/fast-display-mode/silhouette-shown

Specify whether or not the silhouette is shown when interacting with the model.

preferences/graphics/performance/fast-display-mode/status

Specify whether the fast interactive display settings are on, off, or automatic.

preferences/graphics/performance/fast-display-mode/transparency

Specify whether or not transparencies remain when interacting with the model.

preferences/graphics/performance/minimum-frame-rate/

Enter the menu for minimum frame-rate settings.

preferences/graphics/performance/minimum-frame-rate/dynamic-adjustment

Enable/disable dynamic adjustment of quality loss per frame to get to the desired frame rate.

preferences/graphics/performance/minimum-frame-rate/enabled

Enable/disable minimum frame rate.

preferences/graphics/performance/minimum-frame-rate/maximum-culling-threshold

With minimum frame rate enabled, Fluent will not cull beyond this number of pixels.

preferences/graphics/performance/minimum-frame-rate/minimum-culling-threshold

With minimum frame rate enabled, Fluent will cull at least this number of pixels.

preferences/graphics/performance/minimum-frame-rate/target-fps

Specify the target frames-per-second.

preferences/graphics/performance/optimize-for

Choose a preset selection for how graphics are displayed.

preferences/graphics/point-tool-size

Specify the size of the point tool (10-100).

preferences/graphics/show-hidden-surfaces

Enable/disable the display of hidden surfaces.

preferences/icing/

Enter the menu for preferences covering the Fluent Icing workspace.

preferences/icing/advanced-flag

Enable/disable the exposure of advanced settings.

preferences/icing/beta-flag

Enable/disable the exposure of beta features.

preferences/icing/display-mesh-after-case-load

Specify which boundaries are displayed by default after the mesh is read.

preferences/icing/use-launcher

Enable/disable the use of the Fluent Launcher for starting the Fluent solver when connecting to or creating a new **Simulation**. This allows you to launch calculations using a different number of CPUs, a different machine, or a queuing system..

preferences/meshing-workflow/

Enter the menu for preferences covering the Fluent Meshing workflows.

preferences/meshing-workflow/checkpointing_option

Specify how Fluent Meshing will save data when you edit a task.

preferences/meshing-workflow/draw-settings/

Enter the menu for specifying drawing settings.

preferences/meshing-workflow/draw-settings/auto-draw

Enable/disable the automatic display of changes in the graphics window based on the current task.

preferences/meshing-workflow/draw-settings/face-zone-limit

Specify the cutoff number of face zones, beyond which, Fluent Meshing will not automatically display changes.

preferences/meshing-workflow/draw-settings/facet-limit

Specify the cutoff number facets, beyond which, Fluent Meshing will not automatically display changes.

preferences/meshing-workflow/save-checkpoint-files

Enable/disable the saving of task editing data when writing a mesh file.

preferences/meshing-workflow/temp-folder

Specify a temporary location to hold generated mesh files. If nothing is specified, Fluent Meshing will write to `%TEMP%` on Windows and to `/tmp` on Linux.

preferences/meshing-workflow/verbosity

Enable/disable the printing of additional information and messages in the Console.

`preferences/navigation/`

Enter the menu for controlling navigation in Ansys Fluent.

`preferences/navigation/mouse-mapping/`

Enable/disable the printing of additional information and messages in the Console.

`preferences/navigation/mouse-mapping/additional/`

Enter the menu for controlling mouse mappings that include a modifier button such as **Ctrl** and **Shift**.

`preferences/navigation/mouse-mapping/additional/ctrllmbclick`

Specify the action/behavoir for **Ctrl** + left-mouse-button + click.

`preferences/navigation/mouse-mapping/additional/ctrllmbdrag`

Specify the action/behavior for **Ctrl** + left-mouse-button + drag.

`preferences/navigation/mouse-mapping/additional/ctrlmmbclick`

Specify the action/behavior for **Ctrl** + middle-mouse-button + click.

`preferences/navigation/mouse-mapping/additional/ctrlmmbdrag`

Specify the action/behavior for **Ctrl** + middle-mouse-button + drag.

`preferences/navigation/mouse-mapping/additional/ctrlrmbclick`

Specify the action/behavior for **Ctrl** + right-mouse-button + click.

`preferences/navigation/mouse-mapping/additional/ctrlrmbdrag`

Specify the action/behavior for **Ctrl** + right-mouse-button + drag.

`preferences/navigation/mouse-mapping/additional/mouseprobe`

Specify whether the probe action provides a long description or a short description.

`preferences/navigation/mouse-mapping/additional/mousewheel`

Specify the action/behavior of the mouse-wheel.

`preferences/navigation/mouse-mapping/additional/mousewheelsensitivity`

Specify the sensitivity of the mouse-wheel (0 is least sensitive, 1 is most sensitive).

`preferences/navigation/mouse-mapping/additional/reversewheeldirection`

Reverse the behavior of the mouse-wheel.

`preferences/navigation/mouse-mapping/additional/shiftlmbclick`

Specify the action/behavior for **Shift** + left-mouse-button + click.

> > **`preferences/navigation/mouse-mapping/additional/shiftlmbdrag`**
> >
> > Specify the action/behavior for **Shift** + left-mouse-button + drag.
> >
> > **`preferences/navigation/mouse-mapping/additional/shiftmmbclick`**
> >
> > Specify the action/behavior for **Shift** + middle-mouse-button + click.
> >
> > **`preferences/navigation/mouse-mapping/additional/shiftmmbdrag`**
> >
> > Specify the action/behavior for **Shift** + middle-mouse-button + drag.
> >
> > **`preferences/navigation/mouse-mapping/additional/shiftrmbclick`**
> >
> > Specify the action/behavior for **Shift** + right-mouse-button + click.
> >
> > **`preferences/navigation/mouse-mapping/additional/shiftrmbdrag`**
> >
> > Specify the action/behavior for **Shift** + right-mouse-button + drag.

`preferences/`simulation/

Enter the menu for simulation preferences.

> **`preferences/simulation/`flow-model**
>
> Specify the default turbulence model (viscous).
>
> **`preferences/simulation/`local-residual-scaling**
>
> Use local residual scaling for solution residuals.
>
> **`preferences/simulation/`report-definitions/**
>
> Enter the menu for report definition preferences.
>
> **`preferences/simulation/report-definitions/`automatic-plot-file**
>
> New report definitions will automatically create associated report files and plots.

Chapter 10: `report/`

`report/dpm-extended-summary`

Create an extended discrete phase summary report of the discrete phase injection(s). You can choose whether you want to save the extended report to a file or print it in the console window. For unsteady tracking, you will be asked whether you want to include in-domain particle/tracks in the report. You will be also prompted whether you want to select a single injection for the summary report. By default, all injections are included. The output depends on whether you have enabled the `report/dpm-zone-summaries-per-injection?` text command, in which case additional information is printed for escaped particles, such as per-injection data. Note that, for unsteady particle tracking, it is necessary to enable the `report/dpm-zone-summaries-per-injection?` text command before any particle parcels are injected into the domain.

`report/dpm-histogram/`

Enters the DPM histogram menu.

`report/dpm-histogram/`**compute-sample**

Computes the minimum/maximum of a sample variable.

`report/dpm-histogram/`**delete-sample**

Deletes a sample from the loaded sample list.

`report/dpm-histogram/`**list-samples**

Shows all samples in a loaded sample list.

`report/dpm-histogram/`**pick-sample-to-reduce**

Select a sample to be reduced.

`report/dpm-histogram/`**plot-sample**

Plots a histogram of a loaded sample.

`report/dpm-histogram/`**read-sample**

Reads a sample file and adds it to the sample list.

`report/dpm-histogram/`**reduce-picked-sample**

Reduce a sample as specified by the data reduction parameters. This command is available only after you selected the sample using the `pick-sample-to-reduce` text command.

report/dpm-histogram/set/

Enters the settings menu for the histogram.

report/dpm-histogram/set/auto-range?

Automatically computes the range of the sampling variable for histogram plots.

report/dpm-histogram/set/correlation?

Computes the correlation of the sampling variable with another variable.

report/dpm-histogram/set/cumulation-curve?

Computes a cumulative curve for the sampling variable or correlation variable when `correlation?` is specified.

report/dpm-histogram/set/diameter-statistics?

Computes the Rosin Rammler parameters, Sauter, and other mean diameters.

report/dpm-histogram/set/histogram-mode?

Uses bars for the histogram plot or xy-style.

report/dpm-histogram/set/logarithmic?

Enables/disables the use of logarithmic scaling on the abscissa of the histogram.

report/dpm-histogram/set/maximum

Specifies the maximum value of the x-axis variable for histogram plots.

report/dpm-histogram/set/minimum

Specifies the minimum value of the x-axis variable for histogram plots.

report/dpm-histogram/set/number-of-bins

Specifies the number of bins.

report/dpm-histogram/set/percentage?

Uses percentages of bins to be computed.

report/dpm-histogram/set/variable^3?

Uses the cubic of the cumulation variable during computation of the cumulative curve.

report/dpm-histogram/set/weighting?

Uses weighting with additional variables when sorting data into samples.

report/dpm-histogram/set/write-sample

Writes a histogram of a loaded sample into a file.

report/setup-reduction/

Enters the settings menu for the sample data reduction. This command is available only after you selected the sample using the `pick-sample-to-reduce` text command.

report/setup-reduction/all-variables-number-of-bins

Sets the number of bins to be used for all variables in the data reduction.

report/setup-reduction/list-settings

Prints all current sample data reduction settings for all variables in the selected sample in the console.

report/setup-reduction/logarithmic?

Enables/disables the logarithmic scaling for a selected variable.

report/setup-reduction/make-steady-from-unsteady-file?

Enables/disables the unsteady sample reduction into a steady-state injection file. This command is available only for sample files from unsteady particle tracking.

report/setup-reduction/maximum

Sets the maximum value of the range to be considered for a specific variable.

report/setup-reduction/minimum

Sets the minimum value of the range to be considered for a specific variable.

report/setup-reduction/number-of-bins

Sets the number of bins for a specific variable.

report/setup-reduction/reset-min-and-max

Resets the specified `maximum` and `minimum` values for a specific variable.

report/setup-reduction/use-weighting?

Enables/disables the use of weighting in the data averaging in each bin.

report/setup-reduction/weighting-variable

Sets the weighting variable for the data averaging in each bin.

report/dpm-sample

Samples trajectories at boundaries and lines/planes.

report/dpm-sample-output-udf

Allows you to hook a previously loaded `DEFINE_DPM_OUTPUT` UDF for file format specification for sampling of trajectories and VOF-to-DPM lump conversion transcripts.

`report/`dpm-sample-sort-file?

Enables/disables writing of sorted DPM sample files.

`report/`dpm-summary

Prints discrete phase summary report.

`report/`dpm-zone-summaries-per-injection?

Enables/disables calculation of the escaped mass per injection. Note that for unsteady particle tracking, if you want to report the mass of escaped particles per injection, this text command must be enabled before any particles are injected into the domain.

`report/`element-mass-flow

Prints 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.

`report/`fluxes/

Enters the fluxes menu.

`report/fluxes/`film-heat-transfer

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

`report/fluxes/`film-mass-flow

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

`report/fluxes/`heat-transfer

Prints heat transfer rate at boundaries.

`report/fluxes/`heat-transfer-sensible

Prints the sensible heat transfer rate at the boundaries.

`report/fluxes/`mass-flow

Prints mass flow rate at inlets and outlets.

`report/fluxes/`rad-heat-trans

Prints radiation heat transfer rate at boundaries.

`report/`forces/

Enters the forces menu.

report/forces/pressure-center

Prints the center of pressure on wall zones.

report/forces/wall-forces

Computes the forces along the specified force vector for all wall zones.

report/forces/wall-moments

Computes the moments about the specified moment center for all wall zones.

report/heat-exchanger/

Enters the heat exchanger menu.

report/heat-exchanger/computed-heat-rejection

Prints total heat rejection.

report/heat-exchanger/inlet-temperature

Prints inlet temperature.

report/heat-exchanger/outlet-temperature

Prints outlet temperature.

report/heat-exchanger/mass-flow-rate

Prints mass flow rate.

report/heat-exchanger/specific-heat

Prints fluid's specific heat.

report/modified-setting/

Enter the modified settings menu.

report/modified-setting/modified-setting

Specify which areas of setup will be checked for non-default settings for generating the **Modified Settings Summary** table. The table is displayed tabbed with the graphics window.

report/modified-setting/write-user-setting

Write the contents of the **Modified Settings Summary** table to a file.

report/mphase-summary

Prints summary report for a multiphase case setup.

report/particle-summary

Prints summary report for all current particles.

report/path-line-summary

Prints pathline summary report.

report/print-histogram

Prints a histogram of a scalar quantity.

report/projected-surface-area

Computes the area of the projection of selected surfaces along the x, y, or z axis.

report/reference-values/

Enters the reference value menu.

report/reference-values/area

Sets reference area for normalization.

report/reference-values/compute/

Computes reference values from zone boundary conditions.

report/reference-values/density

Sets reference density for normalization.

report/reference-values/depth

Sets reference depth for volume calculation.

report/reference-values/enthalpy

Sets reference enthalpy for enthalpy damping and normalization.

report/reference-values/length

Sets reference length for normalization.

report/reference-values/list

Lists current reference values.

report/reference-values/pressure

Sets reference pressure for normalization.

report/reference-values/temperature

Sets reference temperature for normalization.

report/reference-values/velocity

Sets reference velocity for normalization.

report/reference-values/viscosity

Sets reference viscosity for normalization.

report/reference-values/yplus

Sets reference yplus for calculation of Yplus Based Heat Transfer Coefficient.

report/reference-values/zone

Sets reference zone.

report/species-mass-flow

Prints 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.

report/summary

Prints the current settings for physical models, boundary conditions, material properties, and solution parameters.

report/surface-integrals/

Enters the surface integral menu.

report/surface-integrals/area

Prints the area of the selected surfaces.

report/surface-integrals/area-weighted-average

Prints area-weighted average of the specified quantity over the selected surfaces.

report/surface-integrals/facet-avg

Prints the facet average of the specified quantity over the selected surfaces.

report/surface-integrals/facet-max

Prints the maximum of the specified quantity over facet centroids of the selected surfaces.

report/surface-integrals/facet-min

Prints the minimum of the specified quantity over facet centroids of the selected surfaces.

report/surface-integrals/flow-rate

Prints the flow rate of the specified quantity over the selected surfaces.

report/surface-integrals/integral

Prints the integral of the specified quantity over the selected surfaces. You can include a wildcard (*) within the surface names.

report/surface-integrals/mass-flow-rate

Prints the mass flow rate through the selected surfaces.

report/surface-integrals/mass-weighted-avg

Prints the mass-averaged quantity over the selected surfaces.

report/surface-integrals/standard-deviation

Prints the standard deviation of the scalar at the facet centroids of the surface.

report/surface-integrals/sum

Prints sum of scalar at facet centroids of the surfaces.

report/surface-integrals/uniformity-index-area-weighted

Prints the area-weighted uniformity index of the specified quantity over the selected surfaces.

report/surface-integrals/uniformity-index-mass-weighted

Prints the mass-weighted uniformity index of the specified quantity over the selected surfaces.

report/surface-integrals/vector-based-flux

Prints the vector-based flux of the specified quantity over the selected surfaces.

report/surface-integrals/vector-flux

Prints the vector flux over the selected surfaces.

report/surface-integrals/vector-weighted-average

Prints the vector-averaged quantity over the selected surfaces.

report/surface-integrals/vertex-avg

Prints the vertex average of the specified quantity over the selected surfaces.

report/surface-integrals/vertex-max

Prints the maximum of the specified quantity over vertices of the selected surfaces.

report/surface-integrals/vertex-min

Prints the minimum of the specified quantity over vertices of the selected surfaces.

report/surface-integrals/volume-flow-rate

Prints the volume flow rate through the selected surfaces.

report/system/

Enters the system menu.

report/system/gpgpu-stats

Prints information about installed general purpose graphical processing units.

report/system/proc-stats

Prints Ansys Fluent process information. This is used to report the memory usage of each of the Ansys Fluent processes.

report/system/sys-stats

System information. This is used to report the CPU configuration of the machines where Ansys Fluent processes have been spawned.

report/system/time-stats

Timer information. This is used to report CPU timings for user and kernel processes and detailed solver timings.

report/uds-flow

Prints list of user-defined scalar flow rate at boundaries.

report/volume-integrals/

Enters the volume integral menu.

report/volume-integrals/mass

Prints total mass of a phase within a selected cell zone.

report/volume-integrals/mass-avg

Prints mass-average of scalar over cell zones.

report/volume-integrals/mass-integral

Prints mass-weighted integral of scalar over cell zones.

report/volume-integrals/maximum

Prints maximum of scalar over all cell zones.

report/volume-integrals/minimum

Prints minimum of scalar over all cell zones.

report/volume-integrals/sum

Prints sum of scalar over all cell zones.

report/volume-integrals/twopisum

Prints sum of scalar over all cell zones multiplied by 2π.

`report/volume-integrals/`volume

Prints total volume of specified cell zones.

`report/volume-integrals/`volume-avg

Prints volume-weighted average of scalar over cell zones.

`report/volume-integrals/`volume-integral

Prints integral of scalar over cell zones.

Chapter 11: `server/`

`server/start-client`

> Start the Ansys Fluent remote visualization client.

`server/start-server`

> Starts the server for the Ansys Fluent remote visualization client.

`server/print-connected-clients`

> Prints the name of the connected client and its IP address to the console.

`server/print-server-address`

> Prints the host address and port number of the server to the console.

`server/shutdown-server`

> Shuts-down the server and disconnects the connected client.

`server/write-or-reset-server-info`

> Allows you to create a new `server_info.txt` file (with any name you specify), which resets the password for connecting to this server session. It does not restart the server.

Chapter 12: `solve/`

`solve/animate/`

Enters the animation menu.

`solve/animate/define/`

Enters the animation definition menu.

`solve/animate/define/define-monitor`

Defines new animation.

`solve/animate/define/edit-monitor`

Changes animation monitor attributes.

`solve/animate/objects/`

Enters the object manipulation menu.

`solve/animate/objects/clear-history`

Clears solution animation object history.

`solve/animate/objects/copy`

Copies solution animation object.

`solve/animate/objects/create`

Creates new solution animation object.

`solve/animate/objects/delete`

Deletes solution animation object.

`solve/animate/objects/edit`

Edits solution animation object.

`solve/animate/playback/`

Enters the animation playback menu.

`solve/animate/playback/delete`

Deletes animation sequence.

solve/animate/playback/play

Plays the selected animation.

solve/animate/playback/read

Reads new animation from file or already defined animations.

solve/animate/playback/set-custom-frames

Specify a custom start frame, increment, and end frame for video export.

solve/animate/playback/stored-view?

Plays the 3D animation sequence using the view stored in the sequence.

solve/animate/playback/video/

Enters the video menu.

solve/animate/playback/video/advanced-quality/

Enters the advanced quality menu.

solve/animate/playback/video/advanced-quality/bitrate-scale

Sets the bitrate scale for saving the video file. The best is 64000 | high is 32000 | medium is 16000 | low is 8000.

solve/animate/playback/video/advanced-quality/enable-h264

Controls whether H.264 encoding is used when the video file is written.

solve/animate/playback/video/format

Specifies what format the video file will be written in (MP4 | AVI | FLV | MOV | MPEG).

solve/animate/playback/video/fps

Sets the target frames per second (FPS) for the saved video file.

solve/animate/playback/video/name

Sets the name of the saved video file.

solve/animate/playback/videoquality

Sets the quality of the saved video file.

solve/animate/playback/video/scale

Expands or collapses the resolution based on this scaling factor.

solve/animate/playback/video/use-original-resolution

Controls whether the resolution of the saved video file matches that of the images used to create it.

solve/animate/playback/write

Writes animation sequence to the file.

solve/**cell-register-operations/**

Enters the cell register operations menu.

solve/cell-register-operations/add

Creates a new cell register operation.

solve/cell-register-operations/delete

Deletes a cell register operation.

solve/cell-register-operations/edit

Edits an existing cell register operation.

solve/cell-register-operations/list

Lists the currently defined cell register operations.

solve/cell-register-operations/list-properties

Lists the properties of a report register operation.

solve/**cell-registers/**

Enters the cell registers menu.

solve/cell-registers/add

Creates a new cell register.

solve/cell-registers/apply-poor-mesh-numerics

Applies poor mesh numerics to the mesh of a cell register.

solve/cell-registers/coarsen

Coarsen the mesh based on a cell register.

solve/cell-registers/delete

Deletes a cell register.

solve/cell-registers/display

Displays a cell register.

solve/cell-registers/edit

Edits an existing cell register.

solve/cell-registers/list

Lists all of the currently defined cell registers.

solve/cell-registers/list-properties

Lists the properties of a cell register.

solve/cell-registers/refine

Refine the mesh based on a cell register.

solve/convergence-conditions/

Enters the convergence conditions menu.

solve/convergence-conditions/condition

Specifies whether the solution is considered converged when all of the conditions are met or when one of the conditions is met.

solve/convergence-conditions/conv-reports/

Enters the convergence reports menu.

solve/convergence-conditions/conv-reports/add

Creates a new convergence condition.

solve/convergence-conditions/conv-reports/delete

Deletes a convergence condition.

solve/convergence-conditions/conv-reports/edit

Edits a convergence condition.

solve/convergence-conditions/conv-reports/list

Lists all of the report definition-based convergence conditions.

solve/convergence-conditions/conv-reports/list-properties

Lists the properties of the specified convergence condition.

solve/convergence-conditions/frequency

Specifies how often convergence checks are performed.

solve/dpm-update

Updates discrete phase source terms.

`solve/`dual-time-iterate

Performs unsteady iterations for a specified number of time steps.

`solve/`execute-commands/

Enters the execute commands menu.

`solve/execute-commands/`add-edit

Adds or edits execute commands.

`solve/execute-commands/`disable

Disables an execute command.

`solve/execute-commands/`enable

Enables an execute command.

`solve/`initialize/

Enters the flow initialization menu.

`solve/initialize/`compute-defaults/

Enters the compute default values menu.

`solve/initialize/compute-defaults/`all-zones

Initializes the flow field with the default values.

`solve/initialize/compute-defaults/`zone

You can select the type of zone from which you want to compute these values. The types of zones available are:

- `axis`

- `degassing`

- `exhaust-fan`

- `fan`

- `fluid`

- `inlet-vent`

- `intake-fan`

- `interface`

- `interior`

- `mass-flow-inlet`

- mass-flow-outlet

- 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

solve/initialize/dpm-reset

Resets discrete phase source terms to zero.

solve/initialize/fmg-initialization

Initializes using the full-multigrid initialization (FMG).

solve/initialize/hyb-initialization

Initializes using the hybrid initialization method.

solve/initialize/init-acoustics-options

Specifies the number of timesteps for ramping of sound sources and re-initializes the acoustics wave equation solution. For the initialized acoustics solution, Fluent reports the current state of the sound sources ramping.

solve/initialize/init-flow-statistics

Initializes unsteady statistics.

solve/initialize/initialize-flow

Initializes the flow field with the current default values.

solve/initialize/lwf-initialization

Deletes wall film particles and initializes wall film variables to zero. This option is available only with the wall-film DPM boundary condition.

solve/initialize/init-turb-vel-fluctuations

Initializes instantaneous velocity field out of steady state RANS results, for use before enabling a scale resolving simulation such as LES.

solve/initialize/list-defaults

Lists default values.

solve/initialize/mp-localized-turb-init/

Enters the menu for localized turbulent flow initialization.

solve/initialize/mp-localized-turb-init/enable?

Enables/disables localized initialization of turbulent flow variables.

solve/initialize/mp-localized-turb-init/turb-init-parameters

Sets values for the turbulent intensity and turbulent viscosity ratio for localized initialization.

solve/initialize/open-channel-auto-init

Opens channel automatic initialization.

solve/initialize/reference-frame

Sets reference frame to absolute or relative.

solve/initialize/repair-wall-distance

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

solve/initialize/set-defaults/

Sets default initial values.

solve/initialize/set-fmg-initialization/

Enters the set full-multigrid for initialization menu. Initial values for each variable can be set within this menu.

solve/initialize/set-hyb-initialization/

Enters the hybrid initialization menu.

solve/initialize/set-hyb-initialization/general-settings

Enters the general settings menu.

solve/initialize/set-hyb-initialization/turbulence-settings

Enters the turbulence settings menu.

solve/initialize/set-hyb-initialization/species-settings

Enters the species-settings menu.

solve/initialize/show-iterations-sampled

Displays the number of iterations covered by the data sampled for steady statistics.

solve/initialize/show-time-sampled

Displays the amount of simulated time covered by the data sampled for unsteady statistics.

solve/initialize/vof-patch-smooth-options/

Enters the vof patch/smooth options menu.

execute-smoothing

Performs volumetric smoothing for volume fraction.

solve/initialize/vof-patch-smooth-options/set-options

Sets options for patching and smoothing volume fraction.

solve/iterate

Performs 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.

solve/mesh-motion

Performs mesh motion.

solve/monitors/

Sets solution monitors.

solve/monitors/residual/

Enters the residual monitors menu.

solve/monitors/residual/check-convergence?

Chooses which currently monitored residuals should be checked for convergence.

solve/monitors/residual/convergence-criteria

Sets convergence criteria for residuals that are currently being both monitored and checked.

solve/monitors/residual/criterion-type

Sets convergence criterion type.

solve/monitors/residual/enhanced-continuity-residual?

Enables/disables an enhanced formulation for the local scaling of the continuity residuals with the pressure-based solver, so that the absolute mass flow rate at each cell is used. This text command is only available when the computing of the local scale is enabled through the `solve/monitors/residual/scale-by-coefficient?` text command.

solve/monitors/residual/monitor?

Chooses which residuals to monitor as printed and/or plotted output.

solve/monitors/residual/n-display

Sets the number of most recent residuals to display in plots.

solve/monitors/residual/n-maximize-norms

Sets the number of iterations through which normalization factors will be maximized.

solve/monitors/residual/normalization-factors

Sets normalization factors for currently monitored residuals (if `normalize?` is set to `yes`).

solve/monitors/residual/normalize?

Chooses whether to normalize residuals in printed and plotted output.

solve/monitors/residual/n-save

Sets number of residuals to be saved with data. History is automatically compacted when buffer becomes full.

solve/monitors/residual/plot?

Chooses whether residuals will be plotted during iteration.

solve/monitors/residual/print?

Chooses whether residuals will be printed during iteration.

solve/monitors/residual/relative-conv-criteria

Sets relative convergence criteria for residuals that are currently being both monitored and checked.

solve/monitors/residual/re-normalize

Re-normalize residuals by maximum values.

solve/monitors/residual/reset?

Chooses whether to delete the residual history and reset iteration counter to 1.

solve/monitors/residual/scale-by-coefficient?

Chooses whether to scale residuals by coefficient sum in printed and plotted output.

solve/monitors/residual/window

Specifies window in which residuals will be plotted during iteration.

solve/patch

Patches a value for a flow variable in the domain. You can either provide a numerical value, which you can enter directly, or you can enter an expression, which must be entered within quotation marks.

solve/report-definitions/

Enters the report definitions menu.

solve/report-definitions/add

Creates a report definition.

solve/report-definitions/copy

Creates a copy of a report definition.

solve/report-definitions/delete

Deletes a report definition.

solve/report-definitions/delete-all

Deletes all of the report definition objects.

solve/report-definitions/edit

Edits a report definition.

solve/report-definitions/list

Lists all defined report definitions.

solve/report-definitions/list-properties

Lists the properties of a report definition.

solve/report-definitions/rename

Renames a report definition.

solve/report-files/

Enters the report files menu.

solve/report-files/add

Creates a report file.

solve/report-files/clear-data

Clears the data associated with a report file.

solve/report-files/delete

Deletes a report file object.

solve/report-files/delete-all

Deletes all of the report file objects.

solve/report-files/edit

Edits a report file.

solve/report-files/list

Lists all defined report files.

solve/report-files/list-properties

Lists the properties of a report file.

solve/report-plots/

Enters the report plots menu.

solve/report-plots/add

Creates a report plot.

solve/report-plots/axes

Defines the axes for a report plot.

solve/report-plots/clear-data

Clears the data associated with a report plot.

solve/report-plots/curves

Defines the curves for a report plot.

solve/report-plots/delete

Deletes a report plot object.

solve/report-plots/delete-all

Deletes all of the report plot objects.

solve/report-plots/edit

Edits a report plot.

solve/report-plots/list

Lists all defined report plots.

solve/report-plots/list-properties

Lists the properties of a report plot.

solve/report-plots/plot

Plots the specified report plot.

solve/rombuilder/

Enters the rombuilder menu.

solve/rombuilder/enable?

Enables/disables the reduced order model.

solve/rombuilder/evaluate

Allows you to enter values between the allowed ranges for the existing variables and zones that are part of the loaded ROM and have the results computed (when a ROMZ file is loaded into Fluent in Workbench).

solve/rombuilder/select-variables-and-zones

Select the variables and locations that you want included in the reduced order model.

solve/set/

Enters the set solution parameters menu.

solve/set/accelerated-non-iterative-time-marching

Enables a modified NITA scheme and other setting changes that can speed up the simulation. This option is only available with the Large Eddy Simulation (LES) turbulence model, and is intended for unreacting flow simulations that use a constant-density fluid.

`solve/set/`acoustics-wave-equation-controls/

Enters the menu to specify parameters of the acoustics wave equation solver.

`solve/set/acoustics-wave-equation-controls/`expert/

Enters the menu to specify the expert parameters.

`solve/set/acoustics-wave-equation-controls/expert/`explicit-relaxation-factor

Specifies the explicit relaxation factor. Should be used only with bad meshes, when iterations do not converge.

`solve/set/acoustics-wave-equation-controls/expert/`under-relaxation-factor

Specifies the implicit under-relaxation factor. Should be used only with bad meshes, when the AMG linear solver does not converge.

`solve/set/acoustics-wave-equation-controls/expert/`max-iterations-per-timestep

Specifies the maximum number if iterations, which will be performed by the acoustics wave equation solver in one timestep.

`solve/set/acoustics-wave-equation-controls/expert/`relative-convergence-criterion

Specifies the iterations convergence criterion in terms of the residual norm drop.

`solve/set/`advanced/

Enters the advanced settings menu.

`solve/set/advanced/`alternate-wall-temp-formulation

Enables/disables an alternate formulation for wall temperatures.

`solve/set/advanced/`bcd-boundedness

Specifies the BCD scheme parameter, which controls the boundedness strength of the BCD scheme in the pressure-based solver, as described in Bounded Central Differencing Scheme in the *Fluent Theory Guide*.

`solve/set/advanced/`bcd-weights-freeze

Enables/disables freezing of weighting coefficients of the central differencing and the upwind components of the BCD scheme. This dialog command requires the iteration number, after which the BCD scheme weights are to be frozen at each timestep. Freezing the BCD weighting coefficients may help to improve convergence of the timestep iterations as described in Bounded Central Differencing Scheme in the *Fluent Theory Guide*.

`solve/set/advanced/`correction-form

Enables/disables the use of the correction form of the discretized momentum equations when using the pressure-based solver, as described in Using the Correction Form of Momentum Discretization in the *User's Guide*.

solve/set/advanced/energy-numerical-noise-filter

Enables/disables a filter to eliminate non-physical numerical noise in the energy field. Numerical noise can appear in solution fields where large variations in specific heat or combustion with phase change are present. Using the energy equation numerical noise filter increases robustness, but may make the solution slightly more diffusive. This text command is only available with the pressure-based solver.

solve/set/advanced/explicit-underrelaxation-value

Sets the explicit under-relaxation value for the density-based implicit solver.

solve/set/advanced/linearized-mass-transfer-udf

Enables/disables the use of a linearized mass transfer user-defined function (that is, the DEFINE_LINEARIZED_MASS_TRANSFER macro), as opposed to a mass transfer user-defined function (that is, the DEFINE_MASS_TRANSFER macro).

solve/set/advanced/retain-cell-residuals

Enables/disables the retention of cell residuals for postprocessing.

solve/set/advanced/retain-temporary-solver-mem

Enables/disables the retention of temporary solver memory, which retains gradient data and makes some advanced options available for postprocessing.

solve/set/advanced/secondary-gradient-limiting/

Enters the secondary gradient limiting menu. This menu is only available when the define/models/solver/pressure-based text command is enabled.

solve/set/advanced/secondary-gradient-limiting/energy?

Enables/disables secondary gradient limiting for the energy calculations on coupled two-sided walls. This limiting can help prevent divergence when the cells on such walls have poor orthogonality.

solve/set/advanced/secondary-gradient-limiting/mesh-quality-limits

Defines the mesh quality limits used when applying secondary gradient limiting to faces on coupled two-sided walls. Shifting this range closer to 1 will decrease the risk of divergence, but at the cost of accuracy.

solve/set/advanced/secondary-gradient-limiting/uds?

Enables/disables secondary gradient limiting for the user-defined scalar (UDS) calculations on coupled two-sided walls. This limiting can help prevent divergence when the cells on such walls have poor orthogonality.

solve/set/advanced/show-all-discretization-schemes

Enables/disables the availability of all applicable discretization schemes.

`solve/set/advanced/singhal-et-al-cavitation-model`

Enables/disables the availability of the Singhal et al. cavitation model option, which can then be enabled in the **Multiphase Model** dialog box or by using the following text command: `define/phases/set-domain-properties/interaction-domain/heat-mass-reaction/cavitation/cavitation?`.

`solve/set/amg-options/`

Enters the AMG options menu

`solve/set/amg-options/aggressive-amg-coarsening?`

Enables / disables the use of a version of the AMG solver that is optimized for high coarsening rates. This option is recommended if the AMG solver diverges with the default settings.

`solve/set/amg-options/amg-gpgpu-options/`

Enters the AMG GPGPU options menu, which contains commands to enable / disable GPGPU acceleration and set solver type and options for individual coupled and scalar equations.

`solve/set/amg-options/conservative-amg-coarsening?`

Enables / disables the use of conservative coarsening techniques for scalar and/or coupled equations that can improve parallel performance and/or convergence for some difficult cases.

`solve/set/amg-options/laplace-coarsening?`

Enables / disables Laplace coarsening for scalar and/or coupled equations.

`solve/set/bc-pressure-extrapolations`

Sets 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 Boundary Acoustic Wave Models 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].

solve/set/convergence-acceleration-for-stretched-meshes/

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

solve/set/correction-tolerance/

Enters the correction tolerance menu.

solve/set/courant-number

Sets the fine-grid Courant number (time step factor). This command is available only for the coupled solvers.

solve/set/data-sampling

Enables data sampling for steady or unsteady flow statistics.

solve/set/data-sampling-options/

Enter the menu for specifying quantities and zones for steady and unsteady flow statistics.

`solve/set/data-sampling-options/add-datasets`

Specify zone and quantity combinations for sampling of steady and unsteady flow statistics.

`solve/set/data-sampling-options/list-datasets`

Lists the combinations of zones and quantities that are defined for sampling steady and unsteady flow statistics.

`solve/set/data-sampling-options/remove-datasets`

Specify which zone and quantity combinations to remove from the sampling of steady and unsteady flow statistics.

`solve/set/disable-reconstruction?`

Completely disables reconstruction, resulting in totally first-order accuracy.

`solve/set/discretization-scheme/`

Enters the discretization scheme menu. This allows you to select the discretization scheme for the convection terms in the solution equations. The following text commands can make a selection from a subset of the models in the following table:

Index	Model
0	First Order Upwind
1	Second Order Upwind
3	Central Differencing
4	QUICK
5	Modified HRIC
6	Third-Order MUSCL
7	Bounded Central Differencing
8	CICSAM
10	Standard
11	Linear
12	Second Order
13	Body Force Weighted
14	PRESTO!
16	Geo-Reconstruct
18	Modified Body Force Weighted
28	Compressive
29	BGM

`solve/set/discretization-scheme/amg-c`

Selects the discretization scheme for the flow equations.

`solve/set/discretization-scheme/epsilon`

Selects the discretization scheme for the turbulent dissipation rate.

`solve/set/discretization-scheme/k`

Selects the discretization scheme for the turbulent kinetic energy.

`solve/set/discretization-scheme/mom`

Selects the discretization scheme for momentum.

`solve/set/discretization-scheme/mp`

Selects which convective discretization scheme for volume fraction is to be used.

`solve/set/discretization-scheme/pressure`

Selects the discretization scheme for pressure.

`solve/set/discretization-scheme/temperature`

Selects the discretization scheme for energy.

`solve/set/divergence-prevention/`

Enters the divergence prevention menu. This menu is only available for the density-based solver.

`solve/set/divergence-prevention/enable?`

Enables a divergence prevention option so that Fluent applies under-relaxation to the variables in select cells where the temperature and/or pressure values are approaching the minimum and/or maximum limits.

`solve/set/enable-output-dp-dt?`

Controls whether the output field variable**dp-dt** will be available for transient simulation post-processing. If you select `no`, pressure fields at the previous time steps will not be stored in memory which reduces memory usage.

`solve/set/equation-ordering`

Sets the order in which the model equations are solved, which can affect the convergence speed when you are using the pressure-based solver. The `standard` method is enabled by default and corresponds to the ordering shown in Figure 24.7: Overview of the Iterative Time Advancement Solution Method For the Segregate Solver and Figure 24.8: Overview of the Non-Iterative Time Advancement Solution Method in the Theory Guide; alternatively, you can select the`op-timized-for-volumetric-expansion` method, which is recommended for flows in which the density is strongly dependent on thermal effects, chemical composition, and so on (such as combustion simulations). This text command is not available for steady simulations and/or when a multiphase model is enabled.

`solve/set/equations/`

Selects the equations to be solved.

solve/set/expert

Sets expert options.

solve/set/fast-transient-settings/

Enters the fast transient settings menu. This menu is only available for transient cases that use the density-based solver.

solve/set/fast-transient-settings/rk2

Allows you to enable the use of a two-stage Runge-Kutta scheme for time integration, or revert to the default multi-stage Runge-Kutta scheme. This text command is only available for transient cases that use the density-based explicit formulation.

solve/set/flow-warnings?

Specifies 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.

solve/set/flux-type

Sets the flux type. Note that for the pressure-based solver, you can enable the `Auto Select` option that automatically selects the flux interpolation type based on the setup of the case.

solve/set/gradient-scheme

Sets gradient options.

solve/set/heterogeneous-stiff-chemistry

Sets the heterogeneous stiff-chemistry solver.

solve/set/high-order-term-relaxation/

Enters the High Order Term Relaxation menu.

solve/set/high-order-term-relaxation/enable?

Enables/disables High Order Term Relaxation.

solve/set/high-order-term-relaxation/options/

High Order Term Relaxation Options.

solve/set/high-order-term-relaxation/options/relaxation-factor

Sets the relaxation factor.

solve/set/high-order-term-relaxation/options/variables/

Selects the variables.

solve/set/high-speed-numerics/

Enters the High Speed Numerics menu.

solve/set/high-speed-numerics/enable?

Enables/disables High Speed Numerics.

solve/set/high-speed-numerics/expert

Sets the level of stabilization used to achieve fast convergence. Enter a number 0-5 corresponding to the Mach number range that best characterizes the flow.

solve/set/high-speed-numerics/visualize-pressure-discontinuity-sensor?

Makes available the Pressure Discontinuity Sensor which is a binary identifier equal to 1 if a cell is in proximity of a pressure discontinuity.

solve/set/limiter-warnings?

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

solve/set/limits

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

solve/set/lock-solid-temperature?

Specifies whether you want to lock (or "freeze") the temperature values for all the cells in solid zones (including those to which you have a 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.

solve/set/max-corrections/

Enters the max-corrections menu.

solve/set/mp-mfluid-aniso-drag

Sets anisotropic drag parameters for the Eulerian multiphase model.

solve/set/mp-reference-density

Sets 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

Index	VOF Equation Discretization	Option
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

solve/set/multi-grid-amg

Sets the parameters that govern the algebraic multigrid procedure.

solve/set/multi-grid-controls/

Sets multigrid parameters and termination criteria.

solve/set/multi-grid-fas

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

solve/set/multi-stage

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

solve/set/multiphase-numerics/

Sets multiphase numerics options.

solve/set/multiphase-numerics/advanced-stability-controls/

Enters the menu for the stability controls for multiphase flows. For more information about the below option, see NITA Expert Options and Default and Stability Controls in the *Fluent User's Guide*.

solve/set/multiphase-numerics/advanced-stability-controls/anti-diffusion/

Enters the anti-diffusion menu. This item is available for VOF cases with the **Interfacial Anti-Diffusion** option enabled.

solve/set/multiphase-numerics/advanced-stability-controls/anti-diffusion/enable-dynamic-strength?

Enables dynamic strength to reduce compression in the direction tangential to the interface.

solve/set/multiphase-numerics/advanced-stability-controls/anti-diffusion/set-dynamic-strength-exponent

Sets the cosine exponent in the dynamic strength treatment in Equation 22.1 in the *Fluent User's Guide*).

solve/set/multiphase-numerics/advanced-stability-controls/anti-diffusion/set-maximum-dynamic-strength

Sets the maximum value of dynamic anti-diffusion strength in Equation 22.1 in the *Fluent User's Guide*).

solve/set/multiphase-numerics/advanced-stability-controls/equation-order/

Enters the equation order menu for homogeneous multiphase flow models.

solve/set/multiphase-numerics/advanced-stability-controls/equation-order/solve-flow-last?

When enabled, solves the flow equation at the end of the iteration. This improves the behavior at the start of new time-step if the solution does not converge properly.

solve/set/multiphase-numerics/advanced-stability-controls/hybrid-nita/

Enters the equation order menu for homogeneous multiphase flow models.

solve/set/multiphase-numerics/advanced-stability-controls/hybrid-nita/initial-outer-iterations

Allows you to change the number of initial time-steps and the number of initial outer iterations to control solution stability.

solve/set/multiphase-numerics/advanced-stability-controls/hybrid-nita/instability-detector/

Enters the menu for the instability detector controls.

solve/set/multiphase-numerics/advanced-stability-controls/hybrid-nita/instability-detector/enable-instability-detector?

Enables/disables the instability detector to deal with possible instability problems. Once this option is enabled, additional instability detection options become available.

solve/set/multiphase-numerics/advanced-stability-controls/hybrid-nita/instability-detector/set-cfl-limit

Sets the Courant number limit for detecting unstable events. This command becomes available once the `enable-instability-detector?` text option has been enabled.

solve/set/multiphase-numerics/advanced-stability-controls/hybrid-nita/instability-detector/set-cfl-type

Selects the CFL number type for detection of an unstable event. This command becomes available once the `enable-instability-detector?` text option has been enabled.

solve/set/multiphase-numerics/advanced-stability-controls/hybrid-nita/instability-detector/set-velocity-limit

Sets the velocity limit for detecting unstable events. This command becomes available once the `enable-instability-detector?` text option has been enabled.

solve/set/multiphase-numerics/advanced-stability-controls/hybrid-nita/instability-detector/unstable-event-outer-iterations

Sets the number of outer iterations for an unstable event.

solve/set/multiphase-numerics/advanced-stability-controls/hybrid-nita/outer-iterations

Sets the number of outer iterations.

solve/set/multiphase-numerics/advanced-stability-controls/p-v-coupling/

Enters the menu for the pressure-velocity coupling controls.

solve/set/multiphase-numerics/advanced-stability-controls/p-v-coupling/coupled-vof/

Enters the stability control menu for VOF cases that involve the Coupled pressure-velocity coupling scheme.

solve/set/multiphase-numerics/advanced-stability-controls/p-v-coupling/coupled-vof/buoyancy-force-linearization?

Allows you to use the linearized buoyancy force and the blended treatment for the buoyancy force.

solve/set/multiphase-numerics/advanced-stability-controls/p-v-coupling/pressure-interpolation/

Enters the pressure interpolation menu.

solve/set/multiphase-numerics/advanced-stability-controls/p-v-coupling/pressure-interpolation/modified-bfw-scheme?

Enables/disables the modified body force weighted interpolation scheme. See Default and Stability Controls in the *Fluent User's Guide* for more information about this option.

solve/set/multiphase-numerics/advanced-stability-controls/p-v-coupling/rhie-chow-flux/

Enters the stability controls menu for the Rhie-Chow interpolation.

solve/set/multiphase-numerics/advanced-stability-controls/p-v-coupling/rhie-chow-flux/low-order-rhie-chow?

Enables/disables the low-order velocity interpolation in the flux calculation.

solve/set/multiphase-numerics/advanced-stability-controls/p-v-coupling/skewness-correction/

Enters the skewness correction menu.

solve/set/multiphase-numerics/advanced-stability-controls/p-v-coupling/skewness-correction/limit-pressure-correction-gradient?

Enables/disables the limited pressure correction gradient in skewness terms for the PISO, SIMPLEC, or fractional step pressure-coupling schemes.

solve/set/multiphase-numerics/advanced-stability-controls/pseudo-transient/

Enters the stability control menu for cases with the steady-state pseudo-transient method.

solve/set/multiphase-numerics/advanced-stability-controls/pseudo-transient/false-time-step-linearization?

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

solve/set/multiphase-numerics/advanced-stability-controls/pseudo-transient/smoothed-density-stabilization-method?

Smooths the cell density near the interface, therefore avoiding unphysical acceleration of the lighter phase in the vicinity of interface. The default number of density smoothings is 2. In case of very large unphysical velocities across the interface, you can increase this number when prompted with `Number of density smoothings`.

solve/set/multiphase-numerics/boiling-parameters/

Enters the menu for the multiphase boiling model parameters.

solve/set/multiphase-numerics/boiling-parameters/liquid-vof-factor

When enabled, considers liquid volume fraction effects by multiplying the heat transfer coefficients by the local liquid volume fraction.

solve/set/multiphase-numerics/boiling-parameters/thin-film

When enabled, includes multiphase boiling thin film effects using Equation 14.505 in the *Fluent Theory Guide*.

solve/set/multiphase-numerics/compressible-flow/

Enters the compressible multiphase flow numerics menu.

solve/set/multiphase-numerics/compressible-flow/alternate-bc-formulation

Enables an alternative formulation for compressible phases at an inlet boundary. This formulation calculates static temperature and pressure using an iterative method based on fundamental thermodynamic relations.

solve/set/multiphase-numerics/compressible-flow/enhanced-numerics

Enables an enhanced numerical treatment that provides better stability at startup and during calculation of compressible flows.

solve/set/multiphase-numerics/default-controls/

Enters the default controls menu. This menu is available only for multiphase flows.

solve/set/multiphase-numerics/default-controls/recommended-defaults-for-existing-cases

Applies the multiphase defaults (version 2020 R1) to the loaded case file.

solve/set/multiphase-numerics/default-controls/undo-r20.1-default-changes?

Allows you to undo multiphase specific default changes (version 2020 R1) applied to the loaded case file.

solve/set/multiphase-numerics/heat-mass-transfer/

Enters the menu for the multiphase heat mass transfer parameters.

solve/set/multiphase-numerics/heat-mass-transfer/alternative-energy-treatment?

Enables the alternative treatment of the energy sources. For more information, see Including Mass Transfer Effects in the *Fluent User's Guide*.

solve/set/multiphase-numerics/heat-mass-transfer/area-density/

Enters the menu for the area density.

solve/set/multiphase-numerics/heat-mass-transfer/area-density/ia-grad-sym?

Enables/disables the interfacial area density Gradient-Symmetric model. For more information about this model, see Algebraic Models in the *Fluent Theory Guide*.

solve/set/multiphase-numerics/heat-mass-transfer/area-density/vof-min-seeding

Sets the minimum volume fraction for the area density and cavitation. This may be useful, for example, in cases when a species mass transfer model (such as the Symmetric model or Particle model) do not consider evaporation or condensation if the volume fraction of one of the phases is zero. The seeding allows for a phase change to occur in the fluid flow. The default value is 1e-6.

solve/set/multiphase-numerics/heat-mass-transfer/boiling/

Enters the menu for the advanced boiling options for the semi-mechanistic boiling model.

solve/set/multiphase-numerics/heat-mass-transfer/boiling/heat-flux-relaxation-factor

Sets the under-relaxation factor for boiling heat flux. See Solution Strategies for the Semi-Mechanistic Boiling Model in the *Fluent User's Guide* for details.

solve/set/multiphase-numerics/heat-mass-transfer/boiling/show-expert-options?

Exposes the expert options for the semi-mechanistic boiling model. For more information about these options, see Expert Options for the Semi-Mechanistic Boiling Model in the *Fluent User's Guide*.

solve/set/multiphase-numerics/heat-mass-transfer/cavitation/

Enters the cavitation heat mass transfer menu.

solve/set/multiphase-numerics/heat-mass-transfer/cavitation/display_clipped_pressure?

Chooses whether to clip pressure contours to the vapor pressure of the fluid wherever the pressure drops below the vapor pressure. By default, pressure contours are clipped.

solve/set/multiphase-numerics/heat-mass-transfer/cavitation/min-vapor-pressure

Sets the minimum vapor pressure limit for the cavitation mass-transfer model. The default value is 1 Pa.

solve/set/multiphase-numerics/heat-mass-transfer/cavitation/max-vapor-pressure-ratio

Sets the maximum limit on the vapor pressure after the turbulence and thermal correction. The default value is five times the vapor pressure, with consideration of turbulent and thermal effects for each cell and phase.

solve/set/multiphase-numerics/heat-mass-transfer/cavitation/schnerr-cond-coeff

Sets the condensation coefficient for the Schnerr-Sauer model (F_{cond} in Equation 14.580 in the *Fluent Theory Guide*). The default and recommended value of 0.2.

solve/set/multiphase-numerics/heat-mass-transfer/cavitation/schnerr-evap-coeff

Sets the evaporation coefficient for the Schnerr-Sauer model (F_{vap} in Equation 14.580 in the *Fluent Theory Guide*). The default and recommended value of 1.

solve/set/multiphase-numerics/heat-mass-transfer/cavitation/turbulent-diffusion

Enables/disables the turbulent diffusion treatment for a cavitating turbulent flow. See Mass Transfer Mechanisms in the *Fluent User's Guide* for details.

solve/set/multiphase-numerics/nita-controls/

Enters the NITA controls menu.

solve/set/multiphase-numerics/nita-controls/face-pressure-options

Sets face pressure options for the pressure calculation. When prompted with `enable body-force-weighted for face pressure calculation?`, enter **yes** to select Body Force Weighted and **no** to select Second Order as a face pressure interpolation method. See Controlling NITA Solution Options via the Text Interface in the *Fluent User's Guide* for details.

solve/set/multiphase-numerics/porous-media/

Enters the porous media numerics menu.

solve/set/multiphase-numerics/porous-media/relative-permeability

Allows you to fix the saturation (volume fraction) of the phase at its user-specified residual saturation value.

solve/set/multiphase-numerics/solution-stabilization/

Enters the solution-stabilization numerics menu. This item is available only for the VOF model.

solve/set/multiphase-numerics/solution-stabilization/additional-stabilization-controls/

Enters the menu for additional advanced stability controls.

solve/set/multiphase-numerics/solution-stabilization/additional-stabilization-controls/blended-compressive-scheme?

Enables/disables the blended compressive discretization scheme.

solve/set/multiphase-numerics/solution-stabilization/additional-stabilization-controls/pseudo-transient-stabilization?

Enables/disables the pseudo-transient momentum stabilization and false time step linearization methods.

solve/set/multiphase-numerics/solution-stabilization/execute-additional-stability-controls?

When enabled, uses additional controls for improved solution stability.

solve/set/multiphase-numerics/solution-stabilization/execute-advanced-stabilization?

When enabled, modifies solver settings for improved solution stability.

solve/set/multiphase-numerics/solution-stabilization/additional-stabilization-controls/execute-settings-optimization?

When enabled, uses optimized settings for VOF.

solve/set/multiphase-numerics/solution-stabilization/velocity-limiting-treatment/

Enters the menu for the velocity-limiting-treatment.

solve/set/multiphase-numerics/solution-stabilization/velocity-limiting-treatment/enable-velocity-limiting?

Enables/disables the velocity limiting treatment.

solve/set/multiphase-numerics/solution-stabilization/velocity-limiting-treatment/set-velocity-cutoff

Specifies the maximum velocity magnitude.

solve/set/multiphase-numerics/solution-stabilization/velocity-limiting-treatment/verbosity?

When enabled, the solver prints the number of velocity limited cells at each iteration.

solve/set/multiphase-numerics/viscous-flow

Enters the viscous multiphase flow numerics menu.

solve/set/multiphase-numerics/viscous-flow/viscosity-averaging

Forces harmonic averaging of cell viscosities to calculate face viscosity used in momentum equation. This can improve convergence for highly viscous flow applications (the VOF model only).

solve/set/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).

solve/set/nita-expert-controls/

Enters the NITA expert control menu.

solve/set/nita-expert-controls/hybrid-nita-settings

Enables and sets hybrid NITA options. For more details, see User Inputs in the *Fluent User's Guide*.

`solve/set/nita-expert-controls/set-verbosity`

Setting this to 1, enables the verbosity for NITA diagnostics. The default value of 0 disables verbosity output for NITA diagnostics.

`solve/set/nita-expert-controls/skewness-neighbor-coupling`

Enables/disables coupling of the neighbor and skewness corrections.

`solve/set/number-of-iterations`

Sets the number of iterations for a steady-state simulation without starting the calculation.

`solve/set/numerical-beach-controls`

Sets 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

`solve/set/numerics`

Sets numerics options.

`solve/set/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 run time.

`solve/set/open-channel-wave-options/`

Sets buffer layer height, verbosity, and open channel wave theory formulation.

`solve/set/open-channel-wave-options/set-buffer-layer-ht`

Sets the buffer layer height.

`solve/set/open-channel-wave-options/set-verbosity`

Sets the open channel wave verbosity.

`solve/set/open-channel-wave-options/stokes-wave-variants`

Specifies which open channel wave theory formulation Fluent uses.

`solve/set/overset/`

Specifies overset meshing solver options.

> ### `solve/set/overset/`**high-order-pressure?**
>
> Uses the pressure gradient of the donor cell in the interpolation of pressure for its receptor cell.
>
> ### `solve/set/overset/`**interpolation-method**
>
> Selects the interpolation method for overset interfaces. Note that the least squares method is recommended for sliding mesh cases.
>
> ### `solve/set/overset/`**orphan-cell-treatment?**
>
> Enables/disables a numerical treatment that attempts to assign reasonable data values to orphan cells.

`solve/set/`**p-v-controls**

Sets pressure-velocity controls.

`solve/set/`**p-v-coupling**

Selects which pressure-velocity coupling scheme is to be used. Five schemes are available:

Index	Model
20	SIMPLE
21	SIMPLEC
22	PISO
24	Coupled
25	Fractional Step

`solve/set/`**phase-based-vof-discretization**

Sets phase based slope limiter for VOF compressive scheme.

`solve/set/`**poor-mesh-numerics/**

Enters the poor mesh numerics menu.

> ### `solve/set/poor-mesh-numerics/`**cell-quality-based?**
>
> Enables/disables the application of poor mesh numerics on cells with an orthogonal quality that is equal to or less than the threshold defined by the `solve/set/poor-mesh-numerics/set-quality-threshold` text command (which by default is set to 0.05).
>
> ### `solve/set/poor-mesh-numerics/`**enable?**
>
> Enables/disables the application of poor mesh numerics on cells (those with an orthogonal quality of 0, as well as those identified by other enabled criteria), and defines whether the local solution correction is 0th, 1st, or 2nd order.

`solve/set/poor-mesh-numerics/`gradient-quality-based?

Enables/disables the detection and treatment of poor cells using a criterion based on the cell gradient quality. Poor mesh numerics are applied when the criterion value is equal to or less than a threshold value defined as part of this text command. This criterion is only available with the pressure-based solver.

`solve/set/poor-mesh-numerics/`print-poor-elements-count

Prints out a listing of the poor cells for each criterion: default, cell quality, and user-defined, and (if enabled) cell gradient quality and solution and cell quality.

`solve/set/poor-mesh-numerics/`reset-poor-elements?

Resets the list of poor cells included by the default, cell quality, user-defined, cell gradient quality, and solution and cell quality criteria.

`solve/set/poor-mesh-numerics/`set-quality-threshold

Sets the orthogonal quality threshold used for applying poor mesh numerics when the `solve/set/poor-mesh-numerics/cell-quality-based?` text command is enabled. By default, cells with an orthogonal quality of 0.05 or lower are corrected.

`solve/set/poor-mesh-numerics/`solution-and-quality-based?

Enables/disables the detection and treatment of poor cells using a criterion based on the solution and cell quality. Poor mesh numerics are applied when the criterion value is equal to or less than a threshold value defined as part of this text command, and at a specified frequency (of iterations or time steps). This criterion is only available with the pressure-based solver.

`solve/set/poor-mesh-numerics/`user-defined-on-register

Includes a register for the poor mesh numerics or not.

`solve/set/`previous-defaults/

Provides text commands that allow you to undo enhancements to the default solver behavior.

`solve/set/previous-defaults/`undo-2019r1-default-changes?

Allows you to undo enhancements introduced in version 2019 R1 of Ansys Fluent, including:

- Maintaining positivity for turbulence variables. These enhancements affect the solution when using an overset interface with the default least squares interpolation method, and may affect convergence when using an ω-based turbulence model or a second-order advection scheme for the turbulence equations. On occasion, these enhancements may have a small adverse effect on performance.

- Using the coupled pseudo-transient solver as a default solver for steady-state, single-phase problems (except for cases that use the battery, fuel cells, melting, or solidification models). If you choose to undo these enhancements, the pressure-velocity coupling scheme will revert to its previous setting.

- Using a more aggressive pseudo time step size for the automatic pseudo time scale method. This enhancement affects the solution when using the coupled pseudo-transient solver and may improve convergence in some cases.

solve/set/previous-defaults/**undo-2019r3-default-changes?**

Allows you to undo enhancements introduced in version 2019 R3 of Ansys Fluent, including:

- Using a formulation for the Rhie-Chow face flux interpolation in the pressure-based solver for single-phase simulations that includes a velocity reconstruction designed to handle non-uniform meshes. For such meshes, this formulation gives more accurate results and prevents non-physical reflections of error waves. Note that for poor quality meshes, while the default under-relaxation factors should work, this formulation may require you to use a smaller under-relaxation factor, a smaller flow Courant number, a smaller pseudo time step size, and/or high order term relaxation (HOTR).

- Using a discretization for poor mesh numerics that enhances robustness for meshes with wall boundary layer resolution (for example, y+=1). This discretization is especially effective at wall surfaces that have high curvature and/or high aspect ratios of the near-wall cells. It is primarily intended to prevent divergence, though it may also improve the solution accuracy.

solve/set/previous-defaults/**undo-2021r1-default-changes?**

Allows you to undo enhancements introduced in version 2021 R1 of Ansys Fluent, including:

- Using the correction form of the discretized momentum equation in the pressure-based solver, which can increase the calculation speed for the double-precision solver (for details on other possible benefits, see Using the Correction Form of Momentum Discretization in the User's Guide). The use of this correction form can also be enabled / disabled through the following text command: `solve/set/advanced/correction-form`.

- Using an enhancement to the warped-face gradient correction, which improves the accuracy of polyhedral and tetrahedral mesh simulations that include a boundary condition with a pressure jump.

- Using a ghost cell approach as part of the gradient limiting process in cells on the axis of 2D axisymmetric swirling flows, in order to improve accuracy.

- Using a weighting to improve the accuracy of gradient calculations in high-aspect-ratio cells adjacent to symmetry faces with the Green-Gauss Node Based gradient method and/or the warped-face gradient correction (WFGC).

- Using an enhancement to the Green-Gauss Node Based gradient method to ensure the consistent application of poor mesh numerics when calculating with different numbers of processes.

- Using the Mach number that is normal to the boundary (rather than the full inflow Mach number) to determine if the flow is supersonic, as part of the specification of the boundary conditions at pressure inlets with the pressure-based solver; this can help resolve convergence issues in some cases.

`solve/set/previous-defaults/`**undo-2021r2-default-changes?**

Allows you to undo the following enhancements introduced in version 2021 R2 of Ansys Fluent:

- Using a gradient computation enhancement for interior mesh interfaces, which resolves convergence issues and may improve accuracy in cases that have interior type mesh interfaces (such as fluid-fluid interfaces) with poor mesh quality.

- Using harmonic averaging of the turbulent kinetic energy for the calculation of the turbulence intensity used by the Intermittency Transition model (γ model) and Algebraic Transition model (Alg-γ), which can help to overcome hysteresis effects (dependence of transition location on initial condition) observed mainly in applications with low turbulence intensity and natural transition.

- Computation of transient field statistics has been slightly changed to now compute the statistics at the end of each time-step rather than at the beginning of each timestep. This corrects some defective behavior that could previously occur with: Using the Extrapolate Variables option together with statistics calculation and statistics computation for certain time-derivative quantities (such as dp/dt). Reverting this change may result in hidden errors.

`solve/set/previous-defaults/`**undo-r19.0-default-changes?**

Allows you to undo enhancements introduced in version 19.0 of Ansys Fluent, including the treatment of symmetry boundary conditions and the treatment of walls with the specified shear condition, as well as an early protection scheme for the linear solver.

`solve/set/`**pseudo-relaxation-factor/**

Enters the pseudo relaxation factor menu.

`solve/set/`**pseudo-transient**

Sets the pseudo transient formulation.

`solve/set/`**pseudo-transient-expert/**

Enters the pseudo transient expert usage control menu.

`solve/set/`**reactions?**

Enables the species reaction sources and sets relaxation factor.

`solve/set/`**relaxation-factor/**

Enters the relaxation-factor menu.

`solve/set/`**relaxation-method**

Sets the solver relaxation method.

`solve/set/`reporting-interval

Sets the number of iterations for which convergence monitors are reported. The default is 1 (after every iteration).

`solve/set/`residual-smoothing

Sets the implicit residual smoothing parameters. This command is available only for the explicit coupled solver.

`solve/set/`residual-tolerance/

Enters the residual tolerance menu.

`solve/set/`residual-verbosity

Sets 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.

`solve/set/`second-order-time-options

Enables / disables the variable time step size formulation for second-order implicit transient formulations. If you disable the variable time step size formulation, note that any change in the time step size will introduce an error proportional to the change in the time step size ratio.

`solve/set/`set-all-species-together

Sets all species discretizations and URFs together.

`solve/set/`set-controls-to-default

Sets controls to default values.

`solve/set/`set-solution-methods-to-default

Sets the solution methods to the default settings.

`solve/set/`set-solution-steering

Sets solution steering parameters.

`solve/set/`slope-limiter-set/

Selects a new Fluent solver slope limiter.

`solve/set/`solution-steering

Enables solution steering for the density-based solver.

`solve/set/`stiff-chemistry

Sets solver options for stiff chemistry solutions.

`solve/set/`surface-tension

Sets surface-tension calculation options.

`solve/set/`transient-controls/

Enters the transient controls menu, which allows you to define settings related to time advancement for transient flow calculations.

`solve/set/transient-controls/`cfl-based-time-stepping

Allows you to specify that an adaptive time stepping method is used in which the time step gets modified by Ansys Fluent as the calculation proceeds such that the Courant–Friedrichs–Lewy (CFL) condition is satisfied, using the specified Courant number.

`solve/set/transient-controls/`duration-specification-method

Sets the method by which you will specify the duration of the calculation. The duration can be defined by the total time, the total number of time steps, the incremental time, or the number of incremental time steps. In this context, "total" indicates that Fluent will consider the amount of time / steps that have already been solved and stop appropriately, whereas "incremental" indicates that the solution will proceed for a specified amount of time / steps regardless of what has previously been calculated. This text command is only available when the time stepping is adaptive or based on a user-defined function.

`solve/set/transient-controls/`error-based-time-stepping

Allows you to specify that an adaptive time stepping method is used in which the time step gets modified by Ansys Fluent based on the specified truncation error tolerance.

`solve/set/transient-controls/`extrapolate-eqn-vars/

Enters the extrapolation menu.

`solve/set/transient-controls/`extrapolate-vars?

Applies a predictor algorithm for computing initial conditions at time step n+1. The predictor algorithm is a computation that sets a better initial condition for the time step.

`solve/set/transient-controls/`fixed-periodic

Allows you to specify that a fixed time stepping method is used in which a specified period or frequency is the basis for determining the time step size and number of time steps.

`solve/set/transient-controls/`fixed-user-specified

Allows you to specify that a fixed time stepping method is used in which you directly specify the time step size and number of time steps.

`solve/set/transient-controls/`multiphase-specific-time-constraints/

Enters the menu for setting multiphase-specific time constraints.

`solve/set/transient-controls/multiphase-specific-time-constraints/mesh-based-cfl-constraint?`

Enables/disables the time step size constraints based on the mesh Courant number.

`solve/set/transient-controls/multiphase-specific-time-constraints/physics-based-constraint?`

Enables/disables the physics-driven time-step constraints.

`solve/set/transient-controls/multiphase-specific-time-constraints/time-scale-options/`

Enters the menu for choosing the time-scale options. This menu becomes available after you enable the physics-driven time-step constraints using the `solve/set/transient-controls/multiphase-specific-time-constraints/physics-based-constraint?` text command.

`solve/set/transient-controls/multiphase-specific-time-constraints/time-scale-options/acoustic-scale?`

Allows you to Include the acoustic time scale. This option is available only for compressible flows.

`solve/set/transient-controls/multiphase-specific-time-constraints/time-scale-options/gravity-scale?`

Allows you to Include the gravitational time scale. This option is available only if gravity is enabled.

`solve/set/transient-controls/multiphase-specific-time-constraints/time-scale-options/surface-tension-scale?`

Allows you to Include the surface-tension-driven time scale. This option is available only if the surface tension force modeling is enabled.

`solve/set/transient-controls/multiphase-specific-time-constraints/time-scale-options/viscous-scale?`

Allows you to Include the viscous time scale. This option is disabled by default. This command is not available with inviscid flows.

`solve/set/transient-controls/multiphase-specific-time-constraints/verbosity?`

When enabled, various time scales (depending on the selection) are printed to the console window at every time step. This text command is available with the moving mesh CFL constraint and with the physics-based constraint.

`solve/set/transient-controls/incremental-time`

Sets the amount of incremental (that is, additional) time to run the simulation, regardless of how much time has already been run in previous calculations. This text command is only

available when the `solve/set/transient-controls/duration-specification-method` is set to 3.

solve/set/transient-controls/max-flow-time

Sets the maximum flow time.

solve/set/transient-controls/max-iterations-per-time-step

Sets the number of time steps for a transient simulation.

> **Note:**
>
> This option is available when automatic initialization and case modification is enabled.

solve/set/transient-controls/multiphase-specific-time-stepping

Allows you to specify that an adaptive time stepping method is used in which the time step gets modified by Ansys Fluent based on the convective time scale (global Courant number): the time-step-size calculation depends on the mesh density and velocity in interfacial cells. This method is available for all multiphase models using the implicit or explicit volume fraction formulation, except for the wet steam model.

solve/set/transient-controls/number-of-time-steps

Sets the number of time steps for a transient simulation without starting the calculation.

solve/set/transient-controls/predict-next-time?

Applies a predictor algorithm for computing the next time step. The predictor algorithm is a computation that sets a better initial condition for the time step. It uses the rate of change between the prediction and the correction as an indicator for whether the next time step should be larger, smaller, or the same as the current one.

solve/set/transient-controls/rotating-mesh-flow-predictor?

Enables / disables an option that allows for better prediction of the flow field in rotating fluid zones at every time step, in order to speed up the calculation. This text command is only available for transient simulations.

solve/set/transient-controls/solid-time-step-size

Allows you to specify that the time step size used for solid zones is independent from that used for fluid zones. This text command is only available when both a solid zone exists and energy is enabled.

solve/set/transient-controls/solution-status

Allows you to open the **Simulation Status** dialog box, which reports details about the simulation.

solve/set/transient-controls/specified-time-step

Specifies whether to define the transient advancement either directly by entering a time step size / period / frequency (using the text commands available in the `solve/set/transient-controls` menu) or indirectly by entering a Courant number value (using the `solve/set/courant-number` text command). This text command is only available for the density-based solver when both the explicit formulation and explicit transient formulation are used.

solve/set/transient-controls/time-step-size

Sets the magnitude of the (physical) time step Δt. This text command is only available when the `solve/set/transient-controls/fixed-user-specified` text command is set to `yes`.

solve/set/transient-controls/time-step-size-for-acoustic-export

Specifies the time interval for acoustic data sampling. This text command is only available when both the Ffowcs Williams and Hawkings model is selected and the density-based solver is used with the explicit formulation and explicit transient formulation.

solve/set/transient-controls/total-number-of-time-steps

Sets the total number of time steps that the simulation will run (which includes any time steps that have already been run in previous calculations). This text command is only available when the `solve/set/transient-controls/duration-specification-method` is set to `1`.

solve/set/transient-controls/total-time

Sets the total amount of time that the simulation will be run (which includes any time that has already been run in previous calculations). This text command is only available when the `solve/set/transient-controls/duration-specification-method` is set to `2`.

solve/set/transient-controls/udf-based-time-stepping

Allows you to specify that the time step size is defined by a user-defined function (UDF) that uses the `DEFINE_DELTAT` macro.

solve/set/transient-controls/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.

solve/set/under-relaxation/

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

`solve/set/`vof-explicit-controls

Sets the sub time step calculation method for VOF calculations.

`solve/set/`vof-numerics

Sets VOF numeric options.

`solve/set/`warped-face-gradient-correction/

Enters the warped-face gradient correction menu.

`solve/set/warped-face-gradient-correction/`enable?

Enables/disables gradient enhancement computations and specifies whether Fluent uses fast or memory saving mode.

`solve/`update-physical-time

Advances 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 13: `surface/`

`surface/`circle-slice

Extracts a circular slice.

`surface/`delete-surface

Removes a defined data surface.

`surface/`edit-surface

Allows you to edit any of the listed available surfaces.

`surface/`imprint-surface

Enters the list of surfaces to imprint from and the name of the new imprinted surface.

`surface/`iso-clip

Clips a data surface (surface, curve, or point) between two isovalues.

`surface/`iso-surface

Extracts an iso-surface (surface, curve, or point) from the current data field.

`surface/`line-slice

Extracts a linear slice in 2D, given the normal to the line and a distance from the origin.

`surface/`line-surface

Defines a "line" surface by specifying the two endpoint coordinates.

`surface/`list-surfaces

Displays the ID and name, and the number of point, curve, and surface facets of the current surfaces.

`surface/`mouse-line

Extracts a line surface that you define by using the mouse to select the endpoints.

`surface/`mouse-plane

Extracts a planar surface defined by selecting three points with the mouse.

`surface/`mouse-rake

Extracts a "rake" surface that you define by using the mouse to select the endpoints.

surface/multiple-iso-surfaces

Creates multiple iso-surfaces at once. Accepts zone names, lists of zone ID's, and wildcards.

surface/multiple-plane-surfaces

Creates multiple planes at once. Accepts zone names, lists of zone ID's, and wildcards.

surface/multiple-zone-surfaces

Creates multiple data surfaces at one time. Accepts zone names, lists of zone ID's, and wildcards.

surface/partition-surface

Defines a data surface consisting of mesh faces on the partition boundary.

surface/plane-slice

Extracts a planar slice.

surface/plane-surface

Create a plane from a coordinate plane, point and normal, or three points.

surface/point-array

Extracts a rectangular array of data points.

surface/point-surface

Defines a "point" surface by specifying the coordinates.

surface/quadric-slice

Extracts a quadric slice.

surface/rake-surface

Extracts a "rake" surface, given the coordinates of the endpoints.

surface/rename-surface

Renames a defined data surface.

surface/reset-zone-surfaces

Recreates missing surface zones by resetting the case surface list.

surface/sphere-slice

Extracts a spherical slice.

surface/structural-point-surface

Defines a structural "point" surface by specifying the coordinates.

surface/surface-cells

Extracts all cells intersected by a data surface.

surface/transform-surface

Transforms surface.

surface/zone-surface

Creates a surface of a designated zone and gives it a specified name.

Chapter 14: `switch-to-meshing-mode`

`switch-to-meshing-mode/`

Switches 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 15: `turbo-post/`

`turbo-post/`**2d-contours**

Displays 2D contours.

`turbo-post/`**avg-contours**

Displays average contours.

`turbo-post/`**compute-report**

Computes turbomachinery quantities.

`turbo-post/`**current-topology**

Sets the current turbo topology for global use.

`turbo-post/`**write-report**

Writes the turbo report to file.

`turbo-post/`**xy-plot-avg**

Displays average XY plots.

Chapter 16: `views/`

`views/`**auto-scale**

Scales and centers the current scene without changing its orientation.

`views/`**camera/**

Enters the camera menu to modify the current viewing parameters.

`views/camera/`**dolly-camera**

Adjusts the camera position and target.

`views/camera/`**field**

Sets the field of view (width and height).

`views/camera/`**orbit-camera**

Adjusts the camera position without modifying the target.

`views/camera/`**pan-camera**

Adjusts the camera target without modifying the position.

`views/camera/`**position**

Sets the camera position.

`views/camera/`**projection**

Toggles between perspective and orthographic views.

`views/camera/`**roll-camera**

Adjusts the camera up-vector.

`views/camera/`**target**

Sets the point to be the center of the camera view.

`views/camera/`**up-vector**

Sets the camera up-vector.

views/camera/zoom-camera

Adjusts 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).

views/default-view

Resets view to front and center.

views/delete-view

Removes a view from the list.

views/last-view

Returns to the camera position before the last manipulation.

views/list-views

Lists predefined and saved views.

views/read-views

Reads views from a view file.

views/restore-view

Uses a saved view.

views/save-view

Saves the current view to the view list.

views/write-views

Writes selected views to a view file.

Chapter 17: Battery Model Text Commands

Once the battery model is enabled either in the graphical user interface or using the following text command:

```
define/models/battery-model
```

```
Enable battery model? [no] y
```

you can access the text user interface through the Console window by entering `define/models/battery-model/`. A listing of the various text commands is as follows:

define/models/battery-model/

Enters the battery model menu.

define/models/battery-model/cht-coupling-parameters

Allows you to define the total heat source for each battery active zone for the CHT coupling solution method.

define/models/battery-model/ecm-parameters

Sets ECM model-specific parameters (see Inputs for the Equivalent Circuit Model for definitions of the ECM model parameters).

define/models/battery-model/electric-field-model/

Enters the electric field setup menu.

define/models/battery-model/electric-field-model/conductive-regions

Specifies active conductive regions.

define/models/battery-model/electric-field-model/contact-resistance-regions

Specifies contact resistance regions.

define/models/battery-model/electric-field-model/current-tap

Sets cathode tap.

define/models/battery-model/electric-field-model/voltage-tap

Sets anode tap.

define/models/battery-model/enable-battery-model?

Enables/disables battery model.

define/models/battery-model/fmu-coupling-parameters

Allows you to hook up FMU output parameters to the Ansys Fluent parameters for the FMU-CHT coupling solution method.

define/models/battery-model/model-parameters

Sets battery model options (see Specifying Battery Model Options for definitions of the model general parameters).

define/models/battery-model/newman-parameters

Sets Newman model-specific parameters (see Inputs for the Newman's P2D Model for definitions of the Newman model parameters).

define/models/battery-model/ntgk-parameters

Sets NTGK model-specific parameters (see Inputs for the NTGK Empirical Model for definitions of the NTGK model parameters).

define/models/battery-model/parameter-estimation-tool

Computes the model parameters from battery's testing data. You can use the estimation tool to compute model parameters for the NTGK or ECM model, or for the One-Equation thermal abuse model. You must provide battery's testing data as a text file in a specific format for each model. When you enable this tool and select the model option, Fluent will show the required format for the input file in the console. For more details, see Using Parameter Estimation Tools. You can contact Ansys Technical Support for clarifications on how to use this tool.

define/models/battery-model/solution-method

Sets solution method options (see Specifying Battery Model Options in the *Fluent User's Guide* for details).

define/models/battery-model/thermal-abuse-model

Allows you to choose a thermal abuse model and specify its parameters (see Specifying Advanced Options in the *Fluent User's Guide* for more details).

define/models/battery-model/user-defined-emodel-parameters

Sets-specific parameters for the user-defined model (see Input for the User-Defined E-Model for definitions of the model parameters).

Chapter 18: Fuel Cell Text Commands

This chapter is divided into the following sections:

18.1. Using the PEMFC Text User Interface

All of the features for the PEMFC model that are available through the graphical user interface are also available through text user interface (TUI) commands. The TUI allows text commands to be typed directly in the Ansys Fluent console window where additional information can be extracted and processed for more advanced analysis.

Once the fuel cell module is loaded (see Loading the PEMFC Module), you can access the text user interface through the console window under `define/models/pemfc/`. A listing of the various text commands is as follows:

`define/models/pemfc/advanced-setup/`

Enters the advanced setup menu.

`define/models/pemfc/advanced-setup/list-zones-briefly`

Lists zone names and IDs.

`define/models/pemfc/advanced-setup/contact-resistivity`

Sets contact resistivity.

`define/models/pemfc/advanced-setup/coolant-channel`

Sets coolant channel.

`define/models/pemfc/advanced-setup/stack-management/`

Enters the stack setup menu.

`define/models/pemfc/advanced-setup/stack-management/list-fc-units`

Lists fuel cell units.

`define/models/pemfc/advanced-setup/stack-management/list-zones-briefly`

Lists zone names and IDs.

define/models/pemfc/advanced-setup/stack-management/create-fc-unit

Creates the fuel cell unit.

define/models/pemfc/advanced-setup/stack-management/modify-fc-unit

Modifies the fuel cell unit.

define/models/pemfc/advanced-setup/stack-management/delete-fc-unit

Deletes the fuel cell unit.

define/models/pemfc/advanced-setup/stack-management/set-stack-current-density

Sets the current density on the anode or cathode (in A/cm^2) and modifies the current solution to assist convergence. This item is only available if the case contains valid data (for example, after initialization, iterating, or reading in data). For more information, see IV-Curve Calculations Using the Text Interface (p. 532).

define/models/pemfc/advanced-setup/stack-management/set-stack-voltage

Sets the voltage difference in Volts between the anode and the cathode and modifies the current solution to assist convergence. This item is only available if the case contains valid data (for example, after initialization, iterating, or reading in data). For more information, see IV-Curve Calculations Using the Text Interface (p. 532).

define/models/pemfc/advanced-setup/stack-management/reset-setup

Resets the stack setup in case mistakes are made.

define/models/pemfc/advanced-setup/stack-management/submit-setup

Submits the stack setup and makes the stack setup take effect.

define/models/pemfc/anode-setup/

Enters the anode setup menu.

define/models/pemfc/anode-setup/catalyst-layer

Sets the catalyst layer.

define/models/pemfc/anode-setup/current-collector

Sets current collector.

define/models/pemfc/anode-setup/electrolyte-layer

Sets the electrolyte layer.

define/models/pemfc/anode-setup/flow-channel

Sets the flow channel.

define/models/pemfc/anode-setup/list-zones-briefly

Lists zone names and IDs.

define/models/pemfc/anode-setup/micro-porous-layer

Sets the micro-porous layer.

define/models/pemfc/anode-setup/porous-electrode

Sets porous electrode.

define/models/pemfc/cathode-setup/

Enters the cathode setup menu.

define/models/pemfc/cathode-setup/catalyst-layer

Sets the catalyst layer.

define/models/pemfc/cathode-setup/current-collector

Sets the current collector.

define/models/pemfc/cathode-setup/electrolyte-layer

Sets the electrolyte layer.

define/models/pemfc/cathode-setup/flow-channel

Sets the flow channel.

define/models/pemfc/cathode-setup/list-zones-briefly

Lists zone names and IDs.

define/models/pemfc/cathode-setup/micro-porous-layer

Sets the micro-porous layer.

define/models/pemfc/cathode-setup/porous-electrode

Sets porous electrode.

define/models/pemfc/electrical-tabs-setup

Sets anode and cathode external contacts where electrical potential (UDS0) boundary conditions are applied.

define/models/pemfc/electrolyte-setup/

Enters the electrolyte setup menu.

define/models/pemfc/electrolyte-setup/catalyst-layer

Sets the catalyst layer.

define/models/pemfc/electrolyte-setup/current-collector

Sets current collector.

define/models/pemfc/electrolyte-setup/electrolyte-layer

Sets the electrolyte layer.

define/models/pemfc/electrolyte-setup/flow-channel

Sets the flow channel.

define/models/pemfc/electrolyte-setup/list-zones-briefly

Lists zone names and IDs.

define/models/pemfc/electrolyte-setup/micro-porous-layer

Sets the micro-porous layer.

define/models/pemfc/electrolyte-setup/porous-electrode

Sets porous electrode.

define/models/pemfc/enable-fc-model?

Enables/disables the PEMFC model.

define/models/pemfc/model-options

Sets the PEMFC model options.

define/models/pemfc/model-parameters

Sets the PEMFC model parameters.

define/models/pemfc/reports

Allows you to enable writing data relevant to the fuel cell (cell voltage, mean and minimum values of H_2 mole fractions in anode fluid zones, mean and minimum values of O_2 mole fractions in cathode fluid zones, and total current generated in anode and cathode catalyst zones) to output files and set the monitoring frequency.

18.1.1. IV-Curve Calculations Using the Text Interface

For valid case and data files, there are two text commands available to assist in the IV-curve calculation. These commands are `set-stack-voltage` (aliased as `ssv`) and `set-stack-current-density` (aliased as `ssc`), available from the PEMFC text command menu: `/define/models/pemfc/advanced-setup/stack-management/`.

For fuel cells, you either prescribe the voltage and obtain the total current delivered by the fuel cell as a result, or you specify the total current (via flux boundary conditions multiplied by the area) and obtain the voltage as part of the solution. The details of this IV-relation are specific for each single fuel cell and depend on mass and heat transport, electrochemistry and inlet conditions, outlet condi-

tions, operating conditions, and any other parameter or material property involved in the calculation. The IV-curve is important for applications, because its product is the power delivered by the system.

As described earlier in this manual, you would start a new simulation from fairly static conditions, that is, high voltage/low current (which implies low species transport and low heat generation). After convergence, you typically may be interested in solutions for new electric boundary conditions, that is, either for a new cell/stack voltage or current.

In such cases, simply going to the **Boundary Conditions** task page and changing the value of the electric potential (`uds-0`) boundary condition, typically allows only small changes, most notably for stacks. Otherwise the solution will not converge. This is where the `set-stack-voltage` and `set-stack-current-density` commands are important.

In addition to changing the boundary conditions (either to a prescribed voltage or current density), these commands process the current data in order to estimate the solution for the new boundary conditions. Because these commands modify the data, you are prompted to save your data, if you have not already done so.

Before going into details of the commands, here are some general remarks about electric potential boundary conditions.

For fixed voltage boundary conditions, both external contacts have a fixed value for the electric potential (`uds-0`). The anode value will typically be zero, but it does not have to be. The cathode value will be larger than the anode value and the difference ($V_{cathode} - V_{anode}$) is the positive cell/stack voltage.

For a fixed current boundary condition, one external contact has to have a fixed value and the other flux boundary conditions. As described earlier in the manual, typically, the anode will have a fixed (zero) value, and the cathode will be floating, however, you can also set the cathode to a fixed zero potential, yielding a floating negative anode potential.

The `set-stack-voltage` command sets the effective stack voltage, that is, the difference ($V_{cathode} - V_{anode}$). For fixed voltage boundary conditions for the previous solution, boundary conditions on both boundaries are of type fixed value and then the cathode value will be changed accordingly. In the case of fixed current boundary conditions for the previous solution, the flux boundary condition will be changed to a fixed value boundary condition, and the value adjusted accordingly with respect to the other fixed value boundary condition.

The `set-stack-current-density` command sets the current density on one boundary to the desired value. Note that the input will be in $\frac{A}{cm^2}$, *not* $\frac{A}{m^2}$ as you would normally have to enter in the **Boundary Conditions** task page. The reason for this is that average current densities reported in the text command interface are also in $\frac{A}{cm^2}$, and this makes it easier to choose the conditions you would like to prescribe next. Also, flux boundary conditions entered in the **Boundary Conditions** dialog box would have to have a positive sign on the anode side, and a negative sign on the cathode side. The input for the text interface command is just a positive number, signs are automatically accounted for.

For fixed current boundary conditions for the previous solution, the `set-stack-current-density` command changes the respective flux boundary condition accordingly. In the case of fixed voltage boundary conditions for the previous solution, the cathode side is chosen to be changed from a fixed value to a flux boundary condition with the new flux.

The two commands may be mixed in an IV-curve calculation. For the type of boundary condition setups currently described in this manual, boundary condition changes will consistently happen on the cathode side. However, if anode flux boundary conditions had been chosen initially, switching to fixed voltage boundary conditions by `set-stack-voltage` command and then back to fixed current boundary conditions by the `set-stack-current-density` command will then have flux boundary conditions on the cathode side. In this case, using the `set-stack-current-density` command exclusively will preserve the anode flux boundary condition setting.

18.2. Using the Fuel Cell and Electrolysis Text User Interface

All of the features for the Fuel Cell and Electrolysis Model (sometimes referred to as the Resolved Electrolyte model) that are available through the graphical user interface are also available through text user interface (TUI) commands. The TUI allows text commands to be typed directly in the Ansys Fluent console window where additional information can be extracted and processed for more advanced analysis.

Once the fuel cell module is loaded (see Loading the Fuel Cell and Electrolysis Module), you can access the text user interface through the console window under `define/models/resolved-MEA-fc/`. A listing of the various text commands is as follows:

`define/models/`**resolved-MEA-fuelcells/**

> Fuel cell model menu.

> `define/models/resolved-MEA-fuelcells/`**advanced-setup/**

>> Advanced setup.

>> `define/models/resolved-MEA-fuelcells/advanced-setup/`**contact-resistivity**

>>> Set contact resistivity.

>> `define/models/resolved-MEA-fuelcells/advanced-setup/`**coolant-channel**

>>> Set coolant channel.

>> `define/models/resolved-MEA-fuelcells/advanced-setup/`**list-zones-briefly**

>>> List zone names and IDs.

>> `define/models/resolved-MEA-fuelcells/advanced-setup/`**stack-management/**

>>> Stack setup.

>>> `define/models/resolved-MEA-fuelcells/advanced-setup/stack-management/`**create-fc-unit**

>>>> Create the fuel cell unit.

>>> `define/models/resolved-MEA-fuelcells/advanced-setup/stack-management/`**delete-fc-unit**

>>>> Delete the fuel cell unit.

define/models/resolved–MEA–fuelcells/advanced-setup/stack-manage-ment/list-fc-units

List the fuel cell units.

define/models/resolved–MEA–fuelcells/advanced-setup/stack-manage-ment/list-zones-briefly

List zone names and IDs.

define/models/resolved–MEA–fuelcells/advanced-setup/stack-manage-ment/modify-fc-unit

Modify the fuel cell unit.

define/models/resolved–MEA–fuelcells/advanced-setup/stack-management/re-set-setup

Reset the stack setup in case mistakes are made.

define/models/resolved–MEA–fuelcells/advanced-setup/stack-manage-ment/set-stack-current-density

Set the current density on the anode or cathode and modify the current solution to assist convergence. Note: Input here is in units of A/cm2. This is only available if the case contains valid data (for example, after initialization, iterating, or reading in data). For more information, see .

define/models/resolved–MEA–fuelcells/advanced-setup/stack-manage-ment/set-stack-voltage

Set the voltage difference in Volts between the anode and the cathode and modify the current solution to assist convergence. This is only available if the case contains valid data (for example, after initialization, iterating, or reading in data). For more information, see .

define/models/resolved–MEA–fuelcells/advanced-setup/stack-manage-ment/submit-setup

Submit the stack setup and makes the stack setup take effect.

define/models/resolved–MEA–fuelcells/advanced-setup/stack-manage-ment/suggest-setup

Suggest the stack setup, invoking the automatic stack setup.

define/models/resolved–MEA–fuelcells/anode-setup/

Anode setup.

define/models/resolved–MEA–fuelcells/anode-setup/catalyst-layer

Set catalyst layer.

define/models/resolved-MEA-fuelcells/anode-setup/current-collector

Set current collector.

define/models/resolved-MEA-fuelcells/anode-setup/electrolyte-layer

Set electrolyte layer.

define/models/resolved-MEA-fuelcells/anode-setup/flow-channel

Set flow channel.

define/models/resolved-MEA-fuelcells/anode-setup/list-zones-briefly

List zone names and IDs.

define/models/resolved-MEA-fuelcells/anode-setup/porous-electrode

Set porous electrode.

define/models/resolved-MEA-fuelcells/cathode-setup/

Cathode setup.

define/models/resolved-MEA-fuelcells/cathode-setup/catalyst-layer

Set catalyst layer.

define/models/resolved-MEA-fuelcells/cathode-setup/current-collector

Set current collector.

define/models/resolved-MEA-fuelcells/cathode-setup/electrolyte-layer

Set electrolyte layer.

define/models/resolved-MEA-fuelcells/cathode-setup/flow-channel

Set flow channel.

define/models/resolved-MEA-fuelcells/cathode-setup/list-zones-briefly

List zone names and IDs.

define/models/resolved-MEA-fuelcells/cathode-setup/porous-electrode

Set porous electrode.

define/models/resolved-MEA-fuelcells/electrolyte-setup/

Electrolyte setup.

define/models/resolved-MEA-fuelcells/electrolyte-setup/catalyst-layer

Set catalyst layer.

define/models/resolved-MEA-fuelcells/electrolyte-setup/current-collector

Set current collector.

define/models/resolved-MEA-fuelcells/electrolyte-setup/electrolyte-layer

Set electrolyte layer.

define/models/resolved-MEA-fuelcells/electrolyte-setup/flow-channel

Set flow channel.

define/models/resolved-MEA-fuelcells/electrolyte-setup/list-zones-briefly

List zone names and IDs.

define/models/resolved-MEA-fuelcells/electrolyte-setup/porous-electrode

Set porous electrode.

define/models/resolved-MEA-fuelcells/enable-fc-model?

Enable/disable fuel cell model.

define/models/resolved-MEA-fuelcells/model-options

Model options.

define/models/resolved-MEA-fuelcells/model-parameters

Model parameters.

define/models/resolved-MEA-fuelcells/reports

Set electrolyte project area and external contacts.

define/models/resolved-MEA-fuelcells/select-model

Select model.

define/models/resolved-MEA-fuelcells/set-default

Set default.

18.2.1. IV-Curve Calculations Using the Text Interface

For valid case and data files, there are two text commands available to assist in the IV-curve calculation. These commands are `set-stack-voltage` (aliased as `ssv`) and `set-stack-current-density` (aliased as `ssc`), available from the Fuel Cell and Electrolysis text command menu: `/define/models/resolved-MEA-fc/advanced-setup/stack-management/`.

For fuel cells, you either prescribe the voltage and obtain the total current delivered by the fuel cell as a result, or you specify the total current (via flux boundary conditions multiplied by the area) and obtain the voltage as part of the solution. The details of this IV-relation are specific for each single

fuel cell and depend on mass and heat transport, electrochemistry and inlet conditions, outlet conditions, operating conditions, and any other parameter or material property involved in the calculation. The IV-curve is important for applications, because its product is the power delivered by the system.

As described earlier in this manual, you would start a new simulation from fairly static conditions, that is, high voltage/low current (which implies low species transport and low heat generation). After convergence, you typically may be interested in solutions for new electric boundary conditions, that is, either for a new cell/stack voltage or current.

In such cases, simply going to the **Boundary Conditions** task page and changing the value of the electric potential (`uds-0`) boundary condition, typically allows only small changes, most notably for stacks. Otherwise the solution will not converge. This is where the `set-stack-voltage` and `set-stack-current-density` commands are important.

In addition to changing the boundary conditions (either to a prescribed voltage or current density), these commands process the current data in order to estimate the solution for the new boundary conditions. Because these commands modify the data, you are prompted to save your data, if you have not already done so.

Before going into details of the commands, here are some general remarks about electric potential boundary conditions.

For fixed voltage boundary conditions, both external contacts have a fixed value for the electric potential (`uds-0`). The anode value will typically be zero, but it does not have to be. The cathode value will be larger than the anode value and the difference ($V_{cathode}$- V_{anode}) is the positive cell/stack voltage.

For a fixed current boundary condition, one external contact has to have a fixed value and the other flux boundary conditions. As described earlier in the manual, typically, the anode will have a fixed (zero) value, and the cathode will be floating, however, you can also set the cathode to a fixed zero potential, yielding a floating negative anode potential.

The `set-stack-voltage` command sets the effective stack voltage, that is, the difference ($V_{cathode}$-V_{anode}). For fixed voltage boundary conditions for the previous solution, boundary conditions on both boundaries are of type fixed value and then the cathode value will be changed accordingly. In the case of fixed current boundary conditions for the previous solution, the flux boundary condition will be changed to a fixed value boundary condition, and the value adjusted accordingly with respect to the other fixed value boundary condition.

The `set-stack-current-density` command sets the current density on one boundary to the desired value. Note that the input will be in $\frac{A}{cm^2}$, *not* $\frac{A}{m^2}$ as you would normally have to enter in the **Boundary Conditions** task page. The reason for this is that average current densities reported in the text command interface are also in $\frac{A}{cm^2}$, and this makes it easier to choose the conditions you would like to prescribe next. Also, flux boundary conditions entered in the **Boundary Conditions** dialog box would have to have a positive sign on the anode side, and a negative sign on the cathode side. The input for the text interface command is just a positive number, signs are automatically accounted for.

For fixed current boundary conditions for the previous solution, the `set-stack-current-density` command changes the respective flux boundary condition accordingly. In the case of fixed voltage boundary conditions for the previous solution, the cathode side is chosen to be changed from a fixed value to a flux boundary condition with the new flux.

The two commands may be mixed in an IV-curve calculation. For the type of boundary condition setups currently described in this manual, boundary condition changes will consistently happen on the cathode side. However, if anode flux boundary conditions had been chosen initially, switching to fixed voltage boundary conditions by `set-stack-voltage` command and then back to fixed current boundary conditions by the `set-stack-current-density` command will then have flux boundary conditions on the cathode side. In this case, using the `set-stack-current-density` command exclusively will preserve the anode flux boundary condition setting.

18.3. Using the Solid Oxide Fuel Cell With Unresolved Electrolyte Text User Interface

All of the features for the Solid Oxide Fuel Cell With Unresolved Electrolyte Model that are available through the graphical user interface are also available through text user interface commands.

Once the fuel cell module is loaded (see Loading the Solid Oxide Fuel Cell With Unresolved Electrolyte Module), you can access the text user interface through the Console Window under `sofc-model`. A listing of the various text commands is as follows:

`define/models/`sofc-model/

   SOFC model menu

   `define/models/sofc-model/`enable-sofc-model?

      Enable/disable SOFC model

   `define/models/sofc-model/`model-parameters

      Set model parameters

   `define/models/sofc-model/`electrochemistry

      Set electrochemistry parameters

   `define/models/sofc-model/`anode-interface

      Set fuel cell anode interface

   `define/models/sofc-model/`cathode-interface

      Set fuel cell cathode interface

   `define/models/sofc-model/`tortuosity-interface

      Set fuel cell tortuosity interface

   `define/models/sofc-model/`electric-field-model/

      Electric field model

      `define/models/sofc-model/electric-field-model/`voltage-tap

         Set voltage tap surface

define/models/sofc-model/electric-field-model/current-tap

Set current tap surface

define/models/sofc-model/electric-field-model/conductive-regions

Set conductive regions

define/models/sofc-model/electric-field-model/contact-resistance-regions

Set contact resistance regions

Chapter 19: Magnetohydrodynamics Text Commands

These are the MHD module text commands available under `define/models/`:

`define/models/mhd-model/`b0-scale-factor

Set and apply external magnetic field scale factor.

`define/models/mhd-model/`boundary-conditions/

Define MHD boundary conditions

> **`define/models/mhd-model/boundary-conditions/`list-zones**
>
> List Ansys Fluent zone information.
>
> **`define/models/mhd-model/boundary-conditions/`fluid**
>
> Set fluid zone boundary condition.
>
> **`define/models/mhd-model/boundary-conditions/`solid**
>
> Set solid zone boundary condition.
>
> **`define/models/mhd-model/boundary-conditions/`wall**
>
> Set wall boundary condition.

`define/models/mhd-model/`enable-mhd?

Enable/disable MHD model.

`define/models/mhd-model/`external-b0-field

Set and apply external magnetic field data.

`define/models/mhd-model/`initialize-dpm

Initialize DPM related MHD variables.

`define/models/mhd-model/`initialize-mhd

Initialize MHD model.

`define/models/mhd-model/`mhd-method

Select MHD method.

define/models/mhd-model/reset-b0-field

Reset external magnetic field data.

define/models/mhd-model/solution-control

Set MHD solution control parameters.

Chapter 20: `continuous-fiber/`

All of the features for the continuous fiber module that are available through the graphical user interface are also available through text user interface (TUI) commands. The TUI allows text commands to be typed directly in the Ansys Fluent console window where additional information can be extracted and processed for more advanced analysis.

Once the fiber module is loaded, you can access the text user interface through the Console Window by pressing return in the console (assuming that you are at the top level menu, that is, not in any other menu such as `define/`). A listing of the various text commands is as follows:

`continuous-fiber/`

> Enter the continuous-fiber menu.

> **`continuous-fiber/define/`**

>> Define fiber models and injections.

>> **`continuous-fiber/define/injection`**

>>> Define and modify fiber injections.

>> **`continuous-fiber/define/model`**

>>> Select a fiber model.

> **`continuous-fiber/file/`**

>> Read and write fiber data.

>> **`continuous-fiber/file/read-fiber-data`**

>>> Read in fiber data file.

>> **`continuous-fiber/file/write-binary-fiber-data`**

>>> Write a fiber data file in binary format.

>> **`continuous-fiber/file/write-fiber-data`**

>>> Write a fiber data file in standard format.

> **`continuous-fiber/print`**

>> Print the fiber data of all defined injections into history files.

continuous-fiber/print-xy

Select two fiber variables for plotting into an xy file.

continuous-fiber/solve/

Solver settings for computing fibers.

continuous-fiber/solve/compute

Compute the fibers for all of the injections.

continuous-fiber/solve/initialize

Initialize all of the fibers and injections.

continuous-fiber/solve/set/

Set solution parameters for the fiber model.

continuous-fiber/solve/set/discretization-scheme/

Select the discretization scheme for convection terms.

continuous-fiber/solve/set/discretization-scheme/energy

Choose a discretization scheme for the fiber energy equation.

continuous-fiber/solve/set/discretization-scheme/momentum

Choose a discretization scheme for the fiber momentum equation.

continuous-fiber/solve/set/discretization-scheme/species

Choose a discretization scheme for the fiber species equation.

continuous-fiber/solve/set/equations/

Select the equations that will be solved.

continuous-fiber/solve/set/equations/energy

Specify the energy equation that will be solved.

continuous-fiber/solve/set/equations/momentum

Specify the momentum equation that will be solved.

continuous-fiber/solve/set/equations/species

Specify the species equation that will be solved.

continuous-fiber/solve/set/interaction/

Set parameters for coupled fiber calculations.

`continuous-fiber/solve/set/interaction/coupled-calculations?`

Select whether to couple fiber and fluid flow calculations.

`continuous-fiber/solve/set/interaction/no.-of-fluid-iters-per-fiber-iter`

Set the number of fluid flow iterations per fiber iteration.

`continuous-fiber/solve/set/interaction/underrelaxation`

Set the underrelaxation factor of the fiber source terms in the fluid flow.

`continuous-fiber/solve/set/monitors/`

Set the residual monitors of the fibers.

`continuous-fiber/solve/set/monitors/check?`

Choose which currently-selected equations should be checked for convergence.

`continuous-fiber/solve/set/monitors/convergence-criteria`

Set the criteria to stop the fiber calculation.

`continuous-fiber/solve/set/monitors/relative-residuals?`

Choose whether relative or absolute convergence criteria will be used.

`continuous-fiber/solve/set/no-of-iterations`

Set the number of fiber iterations that will be computed.

`continuous-fiber/solve/set/reporting-interval`

Set the reporting frequency for the fiber model.

`continuous-fiber/solve/set/under-relaxation/`

Set the underrelaxation parameters.

`continuous-fiber/solve/set/under-relaxation/energy`

Specify the underrelaxation factor for the energy equation.

`continuous-fiber/solve/set/under-relaxation/momentum`

Specify the underrelaxation factor for the momentum equation.

`continuous-fiber/solve/set/under-relaxation/species`

Specify the underrelaxation factor for the species equation.