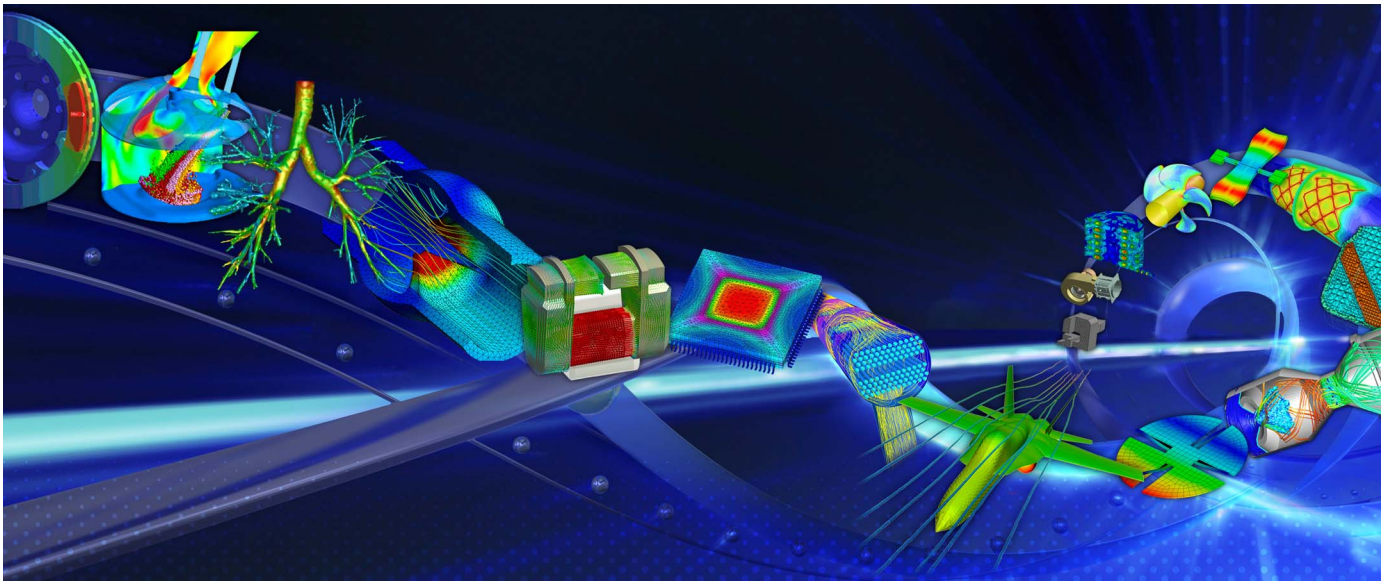


ANSYS CFX-Solver Manager User's Guide

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

Release 12.0
April 2009

Copyright and Trademark Information

Disclaimer Notice

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

ANSYS UK Ltd. is a UL registered ISO 9001:2000 company.

U.S. Government Rights

Third-Party Software

Published in the U.S.A.

Table of Contents

List of Figures

Chapter 1. CFX-Solver Manager Basics

CFX-Solver Manager is a graphical user interface that allows you to set attributes for your CFD calculation, control the CFX-Solver interactively, and view information about the emerging solution. As an alternative to using the CFX-Solver Manager interface, you can also operate CFX-Solver from the command line, which is particularly useful for batch mode operations (see Chapter 11, *Starting the CFX-Solver from the Command Line* (p. 109)).

This chapter describes:

Starting CFX-Solver Manager

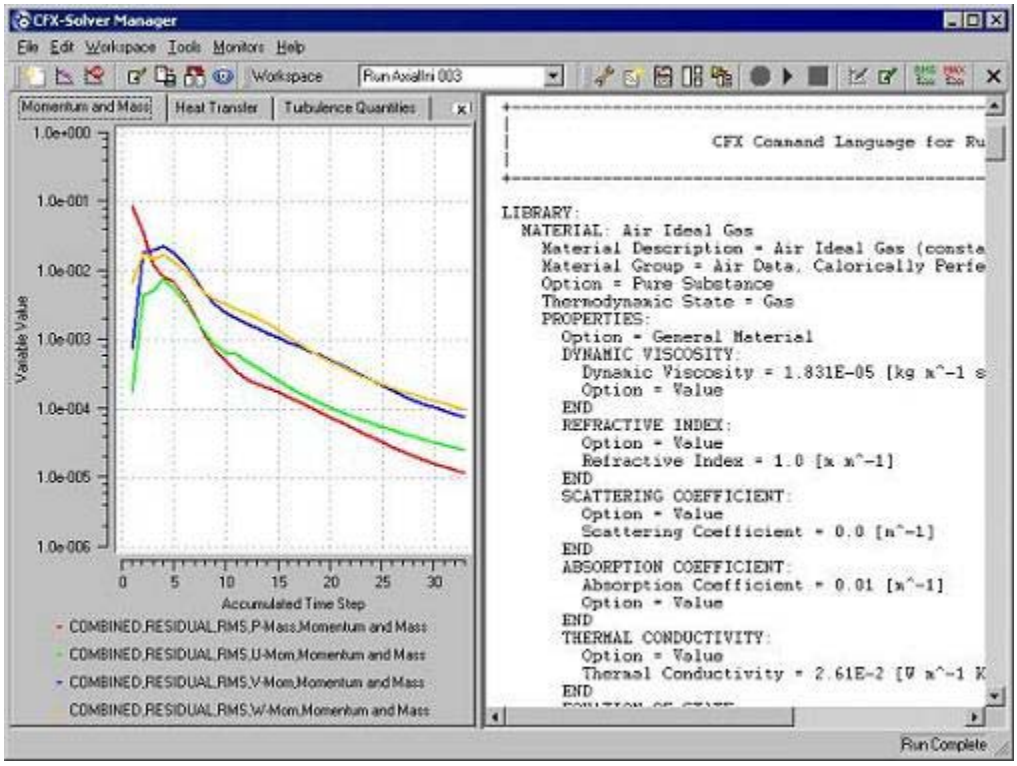
You can start CFX-Solver Manager in different ways:

- Using ANSYS Workbench. For details, see ANSYS CFX in ANSYS Workbench (p. 43) in the ANSYS CFX Introduction.
- Using the CFX-Launcher. For details, see Using the CFX Launcher (p. 41) in the ANSYS CFX Introduction.
- If CFX-Pre is launched and a simulation is open, you can launch CFX-Solver Manager by writing a solver file. For details, see Write Solver Input File Command (p. 46) in the ANSYS CFX-Pre User's Guide.

Working with the CFX-Solver Manager Interface

CFX-Solver Manager is an interface that displays a variety of results as outlined below. It is generally used to view the plotted data during problem solving.

Figure 1.1. CFX-Solver Manager Interface



By default, the convergence history plots appear to the left or the top. The text output window appears to the right or the bottom. There is an adjustable split between the windows that is oriented either horizontally or vertically, depending on the aspect ratio of the entire CFX-Solver Manager window (also adjustable).

The following parts of the interface are described next:

Workspace Selector

The Workspace selector is a drop-down list that shows the current run name and enables you to switch between runs.

1. Click the arrow  of the **Workspace** drop-down list.
2. Select the run you want to view.

Convergence History Plots

Convergence history plots are available on various tabs, with monitors of a certain type under each tab. These are controlled from the *Monitors Menu* (p. 107) and configured from the Monitor Properties (p. 93) dialog (which you access by right-clicking in the plot area).

The main area of convergence history plots shows the plotted value of each variable (typically, an RMS residual) at each timestep, according to the selected monitor (for example, Momentum and Mass, Heat Transfer, etc.). To switch between these views, right-click in the convergence history plot and select **Switch Residual Mode**.

You can edit the plot line variables by selecting **Workspace** > **Workspace Properties** from the main menu. For details, see *Workspace Menu* (p. 91).

You can click any of the plot lines to view the value at the nearest timestep. The legend that appears below each history plot shows the variable associated with each plot line.

Printing an Image of the Convergence History

You can print a picture of the convergence history plot:

1. Right-click in the convergence history plot and select **Print**.
2. Configure the printer as required.
3. Click **Print**.

Saving a Picture of the Convergence History

You can save a picture of the convergence history plot:

1. Right-click in the convergence history plot and select **Save Picture**.

   The **Image File** dialog box is displayed.
2. In the **Image File** dialog box, select a location to which to export the image.
3. Under **File name**, enter the name for the file.
4. Under **File type**, select the format to export.
5. Click **Save**.

Exporting Plot Data

Data from any plot monitor can be exported. The data format for the exported file consists of comma-delimited entries. This data can be used as the basis for import to another application.

1. Right-click in the convergence history plot.
2. Select **Export Plot Data**.

   The **Export** dialog box is displayed.
3. Select a location to which to export the data.

4. Under **File name**, enter the name for the file.
5. Under **File Type**, select the format to export.

   Files are saved with a `.csv` extension by default, but this can be modified as required.
6. Click **Save**.

Note that monitor data can also be exported using the command-line application `cfx5mondata`.

Exporting Monitor Data from the Command Line

The `cfx5mondata` application provides a command-line driven mechanism to query or extract monitor data from the directory in which a solver run is executing, a CFX-Solver Results file, or a monitor file. When extracted, monitor data are written in a comma-separated format that is similar to the output generated by the Exporting Plot Data capability.

To obtain a complete list of available command line options for `cfx5mondata` type the following command into a UNIX terminal or a suitable Windows command line and press **Return** or **Enter**:

```
cfx5mondata -help
```

Text Output Window

The text output window lists simulation information and the progress of a solution, including information such as physical properties, boundary conditions and various other parameters used or calculated in creating the model. All text is written to the output file automatically. For details, see CFX-Solver Output File (p. 28).

For an ANSYS Multi-field run, an additional text output window is created that shows the text output from the ANSYS solver. In this case, you can switch between the CFX output file and the ANSYS Out File by using the tabs at the top of the window. For details on ANSYS Multi-field runs, see Coupling CFX to an External Solver: ANSYS Multi-field Simulations (p. 295).

The right-click menu provides functionality to find, select and copy text, manage bookmarks as well as manage the monitor.

Saving the Text to File

The text of the output window can be saved to file (even when a run is in progress):

1. Right-click in the text output window.
2. Select **Save As**.
3. Select a file location to save the text file.
4. Enter a file name for the text file.
5. Click **Save.**

Searching for Text

The output window can be searched for specific text:

1. Right-click in the text output window and select **Find**.
2. In **Find**, enter words to find.
3. Select or clear the **Case Sensitive** check box.
4. Click **Previous** or **Next** to search up or down from the current location. An icon will appear to the left of the **Find** label if the text could not be found in the file when searching in the selected direction from the current location.

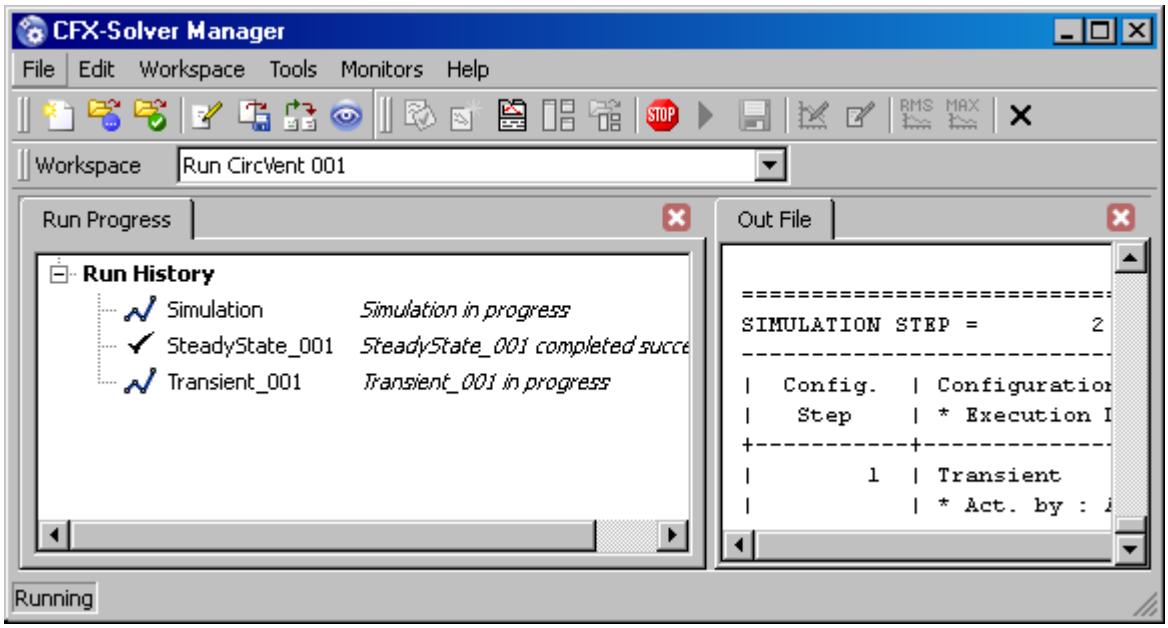
Multi-Configuration Run History Page

A multi-configuration simulation has several workspaces:

- One workspace for each analysis. Each of these workspaces has a convergence history plot and a text output window that resemble those of a single-analysis simulation.

- One workspace that shows an overview of the multi-configuration simulation. There is a run history page that lists each simulation step (analysis) and its overall status (= in progress, = complete, = error), and a text output window that provides the status of each simulation step.

A sample run history page is shown below:



In this case, there are two simulation steps: a steady state analysis and a transient analysis. The steady state analysis (simulation step 1) is complete, and the transient analysis (simulation step 2) is in progress. Because the transient analysis is in progress, the overall simulation is marked as being in progress. When the transient analysis completes, the overall simulation will be marked as being complete.

You can switch to the workspace for a simulation step by right-clicking the simulation step in the run history page and selecting **Open Workspace** from the shortcut menu.

The shortcut menu commands on the run history page are:

- Open Workspace

  This switches to the workspace view for the selected simulation step.

- Post Process Results

  This command is available when running CFX-Solver Manager in standalone mode. It launches CFD-Post with the results of the selected simulation step.

- Post Process Results and Shutdown

  Same as Post Process Results, except CFX-Solver Manager closes after CFD-Post is launched.

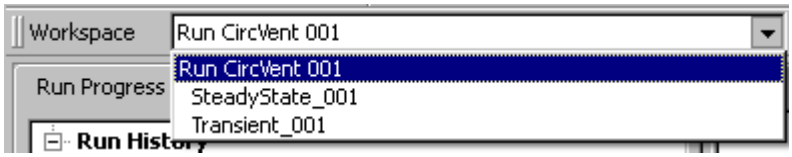
- Stop Run

  This command stops the CFX-Solver from computing the results for the selected simulation step.

- Display Termination Message

  For a simulation step that has completed, this command displays a summary of the termination status.

An alternative way to switch between the various workspaces is to choose a workspace from the **Workspace** drop-down list:



In the drop-down list, the analyses of a multi-configuration simulation are listed below the name of the run, and are indented to show that they belong to the run. Selecting the name of the run switches to the run history page.

Customizing CFX-Solver Manager

The size and position of most windows in CFX-Solver Manager can be changed to customize the appearance, function and placement of objects. Toolbars can also be moved as required; just click and drag the left edge of a toolbar to a new position. (The toolbar orients itself vertically if places along the left or right side of the application.)

Settings such as window sizes, selected plot variables, and other settings specific to the current view can be saved as a layout file (`*.mst`). For example, you could select a plot variable and configure the current view. You could then save the layout and restore it at a later time. For details, see *Workspace Menu* (p. 91).

Chapter 2. Working with Solver Manager

This chapter describes procedures for starting specific types of runs using CFX-Solver Manager. The steps to take depend on whether an initial values file is required or not, and whether the CFX-Solver input file name has already been passed to the CFX-Solver Manager.

This chapter describes:

Solver Run Overview

To run a case, review the settings in the **File** > **Define Run** dialog box. The number of settings that you need to change or specify depends on the case:

- In some cases, you need only to specify the name of a CFX-Solver input file (`*.def` or `*.mdef`). For cases that require initialization from previous results, you also need to specify the name of a results file (`*.res` or `*.mres`).
- You can choose to run in serial or parallel:
  - *Serial run* is the default way of running a CFD case. During a serial run, all computation is done by a single process running on one processor.
  - *Parallel run* divides computation into more than one process and is done on more than one processor in a single machine (local parallel processing) or on more than one machine (distributed parallel processing).
- You can configure an ANSYS Multi-field run, which allows the coupling of the CFX-Solver with the ANSYS Solver in order to execute cases which require two-way fluid-structure interaction.
- For simulations with multiple configurations, you can make global and configuration specific settings. Note that ANSYS Multi-field runs are not supported for multi-configuration simulations.

When you have finished setting up the case, click **Start Run**.

> **Tip**
>
> In ANSYS Workbench you have the option of clicking **Save Settings** to save the settings of the **Define Run** dialog box to the Solution cell (some of these settings are visible in the Solution cell's Properties). By saving the settings in this way, you can configure the run from CFX-Solver Manager, but start the run from ANSYS Workbench (by updating the Solution cell).

Details of making these changes with the **Define Run** dialog box are described in the next section.

The Define Run Dialog Box

You access the **Define Run** dialog box from CFX-Solver Manager by clicking **File** > **Define Run**.

The tabs of the **Define Run** dialog box are described in the following sections:

- Interpolator Tab (p. 12)

Solver Input File Name

Ensure that the name of a CFX-Solver input file (extension `.def` or `.mdef`) is specified under **Solver Input File**.

The name will be set automatically if:

- You have completed the **Write Solver Input File** dialog box in CFX-Pre.
- You have started CFX-Pre from the command line using the `-interactive` and `-def` or `-mdef` options. For details, see *Starting the CFX-Solver from the Command Line* (p. 109).

  CFX-Solver input file names must not contain spaces when run with an associated ANSYS input file (`.inp`).

> **Note**
>
> When specifying or editing the **Solver Input File** by typing from the keyboard, you must click **Reload run settings from file**  to have the changes take effect.

Edit Configuration

If a multi-configuration definition or results file (`*.mdef` or `*.mres`, respectively) is selected as the CFX-Solver Input file, then the settings on the tabs described below can be made on a simulation-wide or per-configuration basis. This is done by choosing either **Global Settings** or a specific configuration name for the **Edit Configuration** option.

> **Note**
>
> When **Global Settings** are made these settings are inherited by all configurations. You can then override any setting for a specific configuration but it is important to note that in this situation the remainder of the settings for the configuration will not be inherited from **Global Settings**.

Run Definition Tab

1. Select **Initial Values Specification** so that you can specify one or more sources of initial values. Note that for cases with multiple configurations, initial values specifications are not valid for Global Settings. For each source of initial values (most runs only require one), do the following:

   a. Click *New*  to create an initial values object.

   b. Select an initial values object from the list and select either the **Results File** or **Configuration Results** option for **Initial Values** > **Option**.

      1. If you selected the **Results File** option, then specify the file name of a file from which initial values should be used.
      2. If you selected the **Configuration Results** option, then specify the name of the configuration from which initial values should be used. Note that this option is only available in the context of multi-configuration simulations. It allows the introduction of dependencies on initial values that will become available at run time.

   c. The **Use Mesh From** setting determines which mesh is used for the analysis: the one specified in the **Solver Input File** option, or the one in the **Initial Values**. The mesh from the Initial Values File can only be used in a limited set of circumstances. See Using the Mesh from the Initial Values File (p. 84) in ANSYS CFX-Solver Modeling Guide for details.

   d. Select **Continue History From** if you want to continue the run history (convergence history, monitor plots, time and time step counters, etc…) and use the smoothest restart possible from the selected Initial Values File. The run will continue from the one contained the specified initial values object. Note that the run history will reset if **Continue History From** is not selected.

   Full details of the settings can be found in Reading the Initial Conditions from a File (p. 81) in ANSYS CFX-Solver Modeling Guide.

2. Set **Type of Run** to `Full` or `Partitioner Only`.
   - `Full` runs the partitioner if applicable, and then runs Solver.
   - `Partitioner Only` is used for parallel runs only and does not run Solver. This writes a `.par` file.
3. Select or clear **Double Precision** or **Executable Selection** > **Double Precision**. This setting will determine the default (single or double) precision of the partitioner, solver and interpolator executables. For details on the precision of executables, see Double-Precision Executables (p. 123). The precision of the solver and interpolator executables can be set individually on the **Solver** and **Interpolator** tabs.
4. Configure the **Parallel Environment** as required.
5. If required, under **Run Environment**, set the working directory.
6. If required, select **Show Advanced Controls** to display other tabs.

Additional information is provided in the next section, *Parallel Environment*, and in Initial Condition Modeling (p. 71) in ANSYS CFX-Solver Modeling Guide.

Parallel Environment

For a distributed parallel setup, specify the number of partitions assigned to each host. If choosing a specified partition weighting (under Partitioner), click directly on the partition weight number to edit it. There should be one weight entry per partition.

1. Under **Parallel Environment**, select a **Run Mode**.
2. Configure the mode as required.

   Run Mode determines whether the run is serial (the default when defining a run in which a problem solved as one process), or parallel (problem split into partitions).
   - Serial run (the default) requires no additional configuration.
   - Parallel Run (p. 14)

MultiField Tab

The **MultiField** tab is used for launching ANSYS Multi-field runs and only appears if the specified CFX-Solver input file has **External Solver Coupling** set to `ANSYS MultiField` or `ANSYS Multifield via Prep7`. When this setting is enabled, the CFX-Solver Manager can be used to launch a complete ANSYS Multi-field run, including processing the multi-field commands (where appropriate) and launching CFX and ANSYS solvers. For details, see Pre-Processing (p. 295) in the ANSYS CFX-Solver Modeling Guide. A description of the different run modes and settings can be found in ANSYS Multi-field Run (p. 18).

The following table outlines various settings available on MultiField tab. The settings are marked as required or optional based on the MFX run mode chosen for the ANSYS Multi-field run.

Settings for MultiField Tab		**MFX Run Mode** [a]			
		Start ANSYS and CFX (default)	**Start ANSYS only**	**Start CFX only** [b]	**Process Input File only**
Input File	Mechanical application input file [c]	Required	Required		Required
	Process ANSYS Input File [d]	Optional	Optional		
Restart ANSYS Run [e] Previous Run DB		Optional	Optional		Optional
MultiField Solver Settings [f] ANSYS Install Directory		Required	Required		

Settings for MultiField Tab	MFX Run Mode [a]			
	Start ANSYS and CFX (default)	Start ANSYS only	Start CFX only [b]	Process Input File only
Additional Arguments ANSYS License				
MultiField Remote Solver Settings Host Name Host Port			Required	

[a]The default setting for **MFX Run Mode** is `Start ANSYS and CFX` that launches the complete MFX run, including starting the ANSYS Solver. The other modes allow you to start just one or other solver, or to process the multi-field commands only using the specified ANSYS Input File. The latter option is available only if the specified CFX-Solver input file has **External Solver Coupling** set to `ANSYS MultiField`.
[b]**Start CFX only**: If the **MFX Run Mode** setting is `Start CFX only`, then no further ANSYS settings are required. However, when the CFX-Solver is started, it needs to know how to communicate with the ANSYS Solver that must have already been started elsewhere (that is, on another machine), and so **Host Name** and **Host Port** must be provided. The host name is the machine on which the ANSYS Solver was started. The host port number is determined by the ANSYS Solver when it starts, and can be read from the file `<jobname>.port` in the ANSYS working directory once the ANSYS Solver has started.
[c]**Mechanical Input File**: For all MFX run modes other than `Start CFX only`, the ANSYS input file must be provided. This will be read from the CFX-Solver input file specified on the **Run Definition** tab; however, you may choose to change this if the file is now in a different location, or if you want to use a different input file.
[d]**Process ANSYS Input File**: For the MFX run modes which will start the ANSYS Solver (`Start ANSYS and CFX` and `Start ANSYS only`), it is necessary to select **Process ANSYS Input File** if the specified ANSYS input file does not already contain the multi-field set-up (the MF commands required for the run), and if you want the multi-field set-up to be read from the CFX-Solver input file. This option will not be available if the specified CFX-Solver input file has **External Solver Coupling** set to `ANSYS MultiField via Prep7`, as in this case the CFX-Solver input file will not contain any multi-field set-up. For details, see Processing the ANSYS Input File (p. 21).
[e]**Restart ANSYS Run**: If the run is a restart from a previous ANSYS Multi-field run, then you need to select **Restart ANSYS Run** and supply the name of the database (`*.db` or `*.rdb`) from the previous run. This is not necessary if you are starting the CFX part of the ANSYS Multi-field run from an existing results file (for example, to provide initial conditions).
[f]**MultiField Solver Settings**: If the ANSYS Solver is to be started, you will need to specify the following settings:

- **ANSYS Install Directory**: On UNIX systems, you may need to manually specify where the ANSYS installation is if it is not in the default location. In this case, you must provide the path to the `v120/ansys` directory.
- **Additional Arguments**: You can use this setting if you want to pass the ANSYS Solver additional arguments. Whatever you specify here will be added to the command which starts the ANSYS Solver.
- **ANSYS License**: This setting is available when running CFX-Solver Manager in standalone mode; it used to select which product variable (license) you want the ANSYS Solver to use. When running in ANSYS Workbench, the **ANSYS License** setting is not available. In this case, the license used by the ANSYS solver is selected by ANSYS Workbench. The latter selects licenses according to the settings found under **Tools** > **Options** > **Licensing** > **License Management**. For details, see the ANSYS Workbench documentation.

Partitioner Tab

Use the **Partitioner** tab to configure the mesh partitioning options.

> **Note**
>
> An existing partition file cannot be used if the simulation involves either the Monte Carlo or Discrete Transfer radiation models.

Partitions may be viewed prior to running CFX-Solver. For details, see CFX Partition File (p. 63).

1. Select the **Partitioner** tab.

   If this is not available, ensure Show Advanced Controls is selected in the **Run Definition** tab.

2. If required, under **Initial Partition File**, click *Browse*  and select the partition file to load.

   The `*.par` file is only required if a model has already been partitioned. The number of partitions in the partitioning file must be the same as that selected on the Run Definition tab.

   > **Note**
   >
   > A partition file generated in ANSYS CFX 11.0 or earlier versions is not supported in ANSYS CFX 12.0. If such a file is used in ANSYS CFX 12.0, then an error message is generated.

3. Under **Run Priority**, select `Idle`, `Low`, `Standard` or `High`. For a discussion of these priorities, see *The cfx5control Application* (p. 129).
4. If required, select the **Use Large Problem Partitioner** option, which is available on 64-bit platforms only. This option starts the large problem partitioner which can partition problems up to 2^31-1 elements. This partitioner uses 64-bit integer and logical variables so it will allocate more memory than the default partitioning executable. For details, see Large Problem Partitioner Executables (p. 123).
5. Under **Partitioning Detail**, choose a **Partition Type** and configure it.

   Depending on the selected partition type, various options must be configured. Partition types include:

   - Multilevel Graph Partitioning Software - MeTiS (p. 347) in the ANSYS CFX-Solver Modeling Guide. When first running in parallel, it is recommended that **Partition Type** be set to **MeTiS**.
   - Recursive Coordinate Bisection (p. 347) in the ANSYS CFX-Solver Modeling Guide
   - Optimized Recursive Coordinate Bisection (p. 347) in the ANSYS CFX-Solver Modeling Guide
   - Directional Recursive Coordinate Bisection (p. 348) in the ANSYS CFX-Solver Modeling Guide
   - User Specified Direction (p. 348) in the ANSYS CFX-Solver Modeling Guide
   - Simple Assignment (p. 347) in the ANSYS CFX-Solver Modeling Guide
   - Radial (p. 349) in the ANSYS CFX-Solver Modeling Guide
   - Circumferential (p. 349) in the ANSYS CFX-Solver Modeling Guide
   - Junction Box (p. 349) in the ANSYS CFX-Solver Modeling Guide
6. If required, configure the **Partition Weighting** as described below.
7. If required, configure the **Multidomain Option**. You can select from the following options:

   - `Independent Partitioning`: Each domain is partitioned independently into the specified number of partitions.
   - `Coupled Partitioning`: All domains that are connected together are partitioned together. Note that solid domains are still partitioned separately from fluid/porous domains. Coupled partitioning often leads to better scalability, reduced memory requirements, and sometimes better robustness, than independent partitioning because there are fewer partition boundaries.

     For details, see Selection of the partitioning mode for multi-domain cases (p. 359) in ANSYS CFX-Solver Modeling Guide.

     When the coupled partitioning option is activated, you can further choose to set the **Multipass Partitioning** option. The `Transient Rotor Stator` option is relevant only for simulations having transient rotor stator interfaces. It uses a special multipass algorithm to further optimize the partition boundaries. This approach generates circumferentially-banded partitions adjacent to each transient rotor stator interface, which ensures that interface nodes remain in the same partition as the two domains slide relative to each other. Away from the interface, the partitioning is handled using whichever method is specified for the **Partition Type**.
8. If required, under **Partitioner Memory**, adjust the memory configuration. For details, see Configuring Memory for the CFX-Solver (p. 13).

Partitioning Weighting

As discussed below, partitions can be weighted in different ways. The default setting is Automatic.

- Uniform
- Specified
- Automatic

Uniform

Assigns equal-sized partitions to each process.

Specified

Requires **Run Definition** to be configured with individual partition weights.

Partition Weights is added to the parallel environment. This allows partition weights to be entered. When more than one partition is assigned to any machine, the number of partition weight entries must equal the number of partitions. The partition weight entries should be entered as a comma-separated list. For a distributed run like the following:

Host	# of Partitions	Partition Weights
Sys01	1	2
Sys02	2	2, 1.5
Sys03	1	1

`Sys01` is therefore a single partition and the weight is 2. `Sys02` has two partitions and they are individually weighted at 2 and 1.5. The final system has a single partition with a weight of 1.

If partition weight factors are used, the ratio of partition weights assigned to each partition controls the partition size.

Once started, the run progresses through the partitioning, and then into the solution of the CFD problem. Extra information is stored in the CFX output file for a parallel run. For details, see Partitioning Information (p. 46).

Automatic

Calculates partition sizes based on the **Relative Speed** entry specified for each machine in the `hostinfo.ccl` file.

Machines with a faster relative speed than others are assigned proportionally larger partition sizes. The entry of relative speed values is usually carried out during the CFX installation process, and accurate entries for relative speed can significantly optimize parallel performance.

Solver Tab

1. Select the **Solver** tab.

   If this is not available ensure Show Advanced Controls on the **Run Definition** tab is selected.

2. Under **Run Priority**, select `Idle`, `Low`, `Standard` or `High`. For a discussion of these priorities as well as how you can change them after the execution of the solver has started, see *The cfx5control Application* (p. 129).
3. If required, from **Double Precision Override** or **Executable Settings** > **Double Precision Override**, select or clear **Double Precision**. This setting for the solver will override the corresponding specification, if set, on the **Run Definition** tab.

   For details, see Double-Precision Executables (p. 123).

4. If required, under **Solver Memory**, adjust the memory configuration. For details, see Configuring Memory for the CFX-Solver (p. 13).
5. Under **Custom Solver Options**, click *Browse*  and select a custom executable.

   This is done when using a custom solver executable. In addition, any command line arguments that must be supplied to the program can be entered under **Solver Arguments**.

Interpolator Tab

1. Select the **Interpolator** tab.

   If this is not available ensure Show Advanced Controls in **Run Definition** tab is selected.

2. Under **Run Priority**, select `Idle`, `Low`, `Standard` or `High`. For a discussion of these priorities, see *The cfx5control Application* (p. 129).

3. If required, from **Double Precision Override** or **Executable Settings** > **Double Precision Override**, select or clear **Double Precision**. This setting for the interpolator will override the corresponding specification, if set, on the **Run Definition** tab.

   For details, see Double-Precision Executables (p. 123).

4. If required, under **Interpolator Memory**, adjust the memory configuration. For details, see Configuring Memory for the CFX-Solver (p. 13).

Configuring Memory for the CFX-Solver

There may be instances when the CFX-Solver fails due to unsufficient memory. You can determine that the CFX-Solver failed in this way as well as which of the interpolator, partitioner, or flow solver steps failed by reviewing the CFX-Solver Output file. In this case you will be required to adjust the memory configuration of the appropriate CFX-Solver step. The methods available for adjusting memory are:

1. **Memory Allocation Factor**: Use this method to modify the memory allocation for the CFX-Solver step as a whole. For example, a value of 1.05 for **Memory Allocation Factor** increases memory allocation by 5% and a value of 1.1 increases memory allocation by 10%.
2. **Detailed Memory Overrides**: Use this method to adjust **Real Memory**, **Integer Memory**, **Character Memory**, **Double Memory** and **Logical Memory** as required.

   The number of words of memory can be specified, or a memory multiplier can be used. Use a unit of **M** for mega-words, **K** for kilo-words, or **X** for a memory multiplier. For example, 2 X doubles the memory allocation and 15 M means 15 million words of memory. If a value is not specified for a particular type of memory, the value calculated by the Solver is used. If a value is entered, it overrides the automatic estimate made by the Solver.

3. **Catalogue Size Override**: If the solver fails with an `*.out` file containing the error message `*** INSUFFICIENT CATALOGUE SIZE ***`, increase the **Catalogue Size Override** setting to a value above 1 until the solver runs. This parameter has the same syntax as the **Detailed Memory Overrides** above. For example, you can scale the default size by using a scale factor greater than 1, such as 1.05 X.

   For details, see *Starting the CFX-Solver from the Command Line* (p. 109).

Run Output Results

Once the run definition is complete and the run is started, a new workspace is created and the **Workspace** drop down list will contain an entry based on the name of the **Solver Input File**. For example, using the **Solver Input File** named `case.def`, the workspace entry will be similar to `Run case 001`. Note that the integer index is identical to the index used for the text output and result files for the run. For multi-configuration simulations, one additional workspace is created as each new configuration is executed. The **Workspace** drop down list will contain entries based on the names of the configurations

Most workspaces contain a mixture of plot and text output monitors that are updated by the CFX-Solver as the simulation progresses. By default, appropriate monitors are automatically created in the workspace for the particular type of simulation you are running. These include:

- • Plot monitors showing the normalized residuals (which should decrease as the solution progresses) for each equation being solved, plus any user-defined monitor values.
- • A text monitor showing the contents of the CFX-Solver Output file. For details, see CFX-Solver Output File (p. 28).

No plot monitors are generated for the simulation-level workspace (named according to the **Solver Input File**) created for multi-configuration simulations. This workspace includes one text monitor, showing the contents of the CFX Multi-Configuration Output File. For details, see CFX Multi-Configuration Output File (p. 64).

When the CFX-Solver stops running, a dialog is displayed that indicates whether the run completed normally or not. Additional information regarding the reasons for terminating the run are presented in the text output window.

After a run has finished:

- Define a new run by following the earlier procedure. For details, see Define Run Command (p. 85)

- Calculate more timesteps for the original run. For details, see Restarting a Run (p. 16).
- View results in CFD-Post (provided that the CFX-Solver produced a results file and did not fail). For details, see Overview of CFD-Post (p. 1) in the ANSYS CFD-Post User's Guide.
- Print residual plots in the convergence history plots. For details, see Printing an Image of the Convergence History (p. 2).
- Add comments to the saved version of the text in the text output window. For details, see *Starting the CFX-Solver from the Command Line* (p. 109).
- Export results in a format suitable for post-processors other than CFD-Post. For details, see *File Export Utility* (p. 131).
- Quit CFX-Solver Manager by selecting **File** > **Quit**. This does not stop the CFX-Solver calculation. Additionally, CFX-Solver Manager can be reopened at any time. For details, see *File Menu* (p. 85).

The CFX Tutorials describe how to use the CFX-Solver Manager step-by-step for several different cases. If you are a new user, you should try at least the first few of these.

Parallel Run

> **Note**
>
> CFX-Solver can be run in parallel only if an appropriate license has been purchased.

Information on a parallel run is explained in more detail:

Overview

There are several parallel run modes, including PVM (Parallel Virtual Machine) and MPI (Message Passing Interface) communication libraries. Both are simply libraries which allow the flow solver processes to communicate with each other. An open-source version of MPI, MPICH, implemented at Argonne National Labs, allows communication over standard TCP/IP networks, as well as through shared memory. Proprietary, vendor specific versions of MPI are available on some platforms (32-bit Linux, 64-bit Linux and HP-UX) and support a wider array of high speed network devices, as well as Shared Memory communication.

General information on setting up a parallel run and advice on obtaining optimal parallel performance is available. For details, see Using the Solver in Parallel (p. 345) in the ANSYS CFX-Solver Modeling Guide.

Individual machines may need to be configured to run in parallel.

General Procedure

To run CFX-Solver in parallel, the following procedure must be followed:

1. Partition the mesh into the appropriate number of partitions.
2. Run CFX-Solver on the partitioned problem.

These two jobs can be done either as one composite run, or as two separate jobs.

Configuring a Parallel Run

1. Follow the procedure for a serial run.

   For details, see Solver Run Overview (p. 7).

2. Under **Run Mode**, select a parallel method:
   - Local Parallel Setup (p. 15)
   - Distributed Parallel Setup (p. 15)

Local Parallel Setup

Select a local parallel run if running a problem with two or more processors on the local machine.

Any number of partitions between 2 and 512 can be selected. When running the job in the CFX-Solver, the computation is divided into this number of processes. For details, see Partitioner Tab (p. 10). Solver allows further changes. For details, see Solver Tab (p. 12).

1. Select one of the local parallel run modes (for example, `HP MPI Local Parallel` or `MPICH Local Parallel`). Which parallel run modes you can select depends on the hardware and operating system on which you are running.
2. Click *Add Partition*  or *Remove Partition*  to increase or decrease the number of partitions.

   Partitions may need to be configured based on partition weighting. For details, see discussion in Partitioner Tab (p. 10).

Distributed Parallel Setup

Select a distributed parallel option to run a problem on two or more computers.

Overview

To configure a distributed parallel run, a file named hostinfo.ccl must exist in the `<CFXROOT>/config/` CFX directory on the master node and be readable by all users of the software. This file is a database containing information about available nodes and where ANSYS CFX is installed on each of them.

> **Note**
>
> - If running machines as slave nodes, ensure rsh service is installed and working on each of those machines (many Windows machines do not already have an rsh service installed). If running MPICH2 on Windows, additional steps are required. For details, see the installation documentation.
> - MPICH cannot be used with a mixture of UNIX and Windows machines.
> - The cfx5parhosts utility operates on the `hostinfo.ccl` file; in releases prior to Release 12.0, it operated on the `hosts.ccl` file. This change was made necessary by a CCL change.

Selecting Parallel Hosts

Simulations can be run on parallel host machines by selecting any available host for parallel processing.

1. Click *Insert Host*  .

   The **Select Parallel Hosts** dialog box is displayed.
2. Select a host from the list of available machines.
3. Click **Add**.

   Ensure the name of current machine is included.
4. Configure each host as required. For details, see Configuring a Host (p. 15).

Configuring a Host

Each parallel host can be configured independently.

1. In **Parallel Environment**, under **Host Name**, select the host to configure.
2. Click *Add Partition*  or *Remove Partition*  to increase or decrease the number of partitions.

   Partitions may need to be configured based on partition weighting. For details, see discussion in Partitioner Tab (p. 10).

Restarting a Run

A CFX-Solver run that has stopped may be restarted for the following reasons:

- The CFX-Solver stopped prematurely, at the user's request, and now needs to continue running.
- More timesteps are needed to extend the duration of a transient analysis, or more iterations are needed to reach a required level of convergence for a steady state analysis.

Such restarts are particularly useful because they continue the analysis from where it left off, which is often much more efficient than re-running the analysis from the original CFX-Solver Input file.

The following discussions describe restarting simulations from CFX-Solver Results files using the same or modified settings:

- Restart Procedure (p. 16)
- Restart Details (p. 16)

Restart Procedure

1. Select **File** > **Define Run**.
2. Select, under **CFX-Solver Input File**, the CFX-Solver Results file of the previous run.
3. Configure the **Define Run** dialog box as required.
4. Click **Start Run**.

> **Tip**
>
> You may also select **Workspace** > **Restart Run** to restart the analysis presented in the active workspace. For details, see *File Menu* (p. 85).

A new workspace that contains the information relevant for that restart, is created for each restart.

Restart Details

Restarting a run should have little effect on the convergence history and no effect on the final results. Additional information regarding several types of restarts is presented below.

- Runs Using Mesh Adaption (p. 16)
- Runs After Physical Model or Solver Parameter Changes (p. 16)
- Runs after Topology or Mesh Changes (p. 17)
- Multi-Configuration Simulations (p. 17)

Runs Using Mesh Adaption

Restarting a run which uses mesh adaption has no effect on the final results. If the maximum number of adaption steps has been specified, then CFX-Solver determines how many adaption steps were completed in the initial run when determining how many adaption steps remain.

Runs After Physical Model or Solver Parameter Changes

You may change CCL settings before continuing from a previously generated CFX-Solver Results file. This is, however, not handled as a restart. For this case, the previously generated results file is first read into CFX-Pre, settings are modified, and a new CFX-Solver Input file is written. Note that this file contains the updated CCL, and the final mesh and mesh adaption parameters from the previous run. A new run is then defined using:

- The newly generated file as the **Solver Input File**,
- The previously generated results file in an **Initial Values** definition.

> **Tip**
>
> On the details view of **Initialization** in CFX-Pre, set the initial conditions for the variables contained in the old results file to **Automatic**. This will ensure that they will be restarted.

Changing the fundamental physics of an analysis, such as the fluids and/or materials involved, is not recommended. Do not change the reference pressure.

> **Note**
>
> If a run which requires wall scale to be calculated is restarted, and a wall scale-related setting was changed (for example, a free slip wall changed to a no-slip wall), wall scale will not automatically be recalculated if its calculation was terminated in a previous run. To force it to be recalculated, set the expert parameter `ignore solve flag on restart` to true. For details on expert parameters, see CFX-Solver Expert Control Parameters (p. 363) in ANSYS CFX-Solver Modeling Guide.

Additional information on initial conditions is available in the Initialization (p. 123) in the ANSYS CFX-Pre User's Guide. More details about using **Initial Values File** in the **Define Run** dialog box in CFX-Solver Manager are available in Reading the Initial Conditions from a File (p. 81) in ANSYS CFX-Solver Modeling Guide.

Runs after Topology or Mesh Changes

If you make changes such as:

- recreating the mesh with different parameters, mesh controls or inflated boundaries,
- changing the underlying geometry,
- changing the connectivity of the geometry (such as specifying domains or subdomains differently),
- adding a new domain or subdomain. or
- changing the name or location of any boundary condition, including the default boundary condition. Note that changing the type of boundary condition is not treated as a topology change,

then you can continue the run as described in Runs After Physical Model or Solver Parameter Changes (p. 16). However, in this case you must set **Use Mesh From** to `Solver Input File`.

Note that changing the type of boundary condition is not treated as a topology change.

Multi-Configuration Simulations

Multi-configuration simulations may only be restarted from a multi-configuration results (.mres) file. The simulation will continue from the final simulation step contained in the specified results file as follows:

- If all active configurations in that simulation step completed successfully, then the new run will begin with the next simulation step.
- If all active configurations in that simulation step failed to complete successfully, then the new run will attempt to complete the final simulation step and execute the configurations that are still active. This occurs if the simulation is stopped prematurely.

Care is required when CCL changes are made before continuing a multi-configuration simulation. This is because CCL is propagated (that is re-used) from the multi-configuration results (`*.mres`) and most recently created configuration results (`*.res`) files. In particular:

- Global CCL changes (for instance, `LIBRARY`, `SIMULATION CONTROL` contents including `EXECUTION CONTROL`, etc…) must be applied to the multi-configuration results file.
- Configuration specific CCL changes (such as `FLOW` contents) must be applied to the most recently generated configuration results files.

> **Important**
>
> Previously created configuration results (`*.res`) files are required for accurate CCL propagation (as noted above) and to resolve configuration dependent initial values, as defined in Run Definition Tab (p. 225) in ANSYS CFX-Pre User's Guide. Paths to these results files are stored and re-used when restarting multi-configuration simulations, and these paths are relative to the working directory. For example, the

stored path for the first run of the configuration name `Configuration1` corresponding to the multi-configuration results file named `mySim_001.mres` is: `mySim_001/configuration1_001.res`.

Use of relative paths allows the directories (and files) for a multi-configuration simulation to be transferred to another working directory (for example, on another file system) to perform a restart. Restarts are not possible if the required directories (and files) have not been transferred to the desired working directory.

Note

If a restart is performed using a specific configuration's result file (for example, `configuration1_001.res`), then only that configuration will continue execution, but the multi-configuration simulation will not continue.

ANSYS Multi-field Run

A tutorial is provided to demonstrate an ANSYS Multi-field run. For details, see Chapter 23, Oscillating Plate with Two-Way Fluid-Structure Interaction.

Note

ANSYS Multi-field runs can only be launched if ANSYS is installed and licensed.

Information on an ANSYS Multi-field run is available in more detail:

Overview

An ANSYS Multi-field run allows the coupling of CFX-Solver with ANSYS Solver in order to execute cases that require two-way fluid-structure interaction. Such cases are described in detail in Using CFX and the Mechanical Application (p. 76) in the ANSYS CFX Reference Guide and Coupling CFX to an External Solver: ANSYS Multi-field Simulations (p. 295) in the ANSYS CFX-Solver Modeling Guide.

General Procedure

The steps which take place when a full ANSYS Multi-field run (where **MFX Run Mode** is set to `Start ANSYS and CFX`) is launched from the CFX-Solver Manager are listed below.

1. User supplies CFX-Solver input file and ANSYS input file.
2. If the ANSYS input file does not already contain multi-field settings, then **Process ANSYS Input File** must be enabled, and a new "multi-field" file with extension `.mf` is created in the ANSYS working directory. This processing step is described more fully in Processing the ANSYS Input File (p. 21).
3. The ANSYS Solver is launched, reading input from the new `.mf` file if it was created in the previous step, or from the original ANSYS input file if that file already contained the multi-field settings.
4. CFX-Solver is launched and is already set up to communicate with ANSYS Solver. The full range of parallel and other solver options (such as double precision) is available for CFX-Solver when run as part of an ANSYS Multi-field run.
5. The run executes.

Many of these steps can be performed separately if required. The workflow and reasons for doing this are described in the following sections:

If a run is to be restarted, a different procedure applies. This described in Restart Procedure for ANSYS Multi-field Runs (p. 22).

Starting ANSYS CFX and ANSYS Separately

The normal procedure for launching an ANSYS Multi-field run from CFX-Solver Manager launches both CFX-Solver and ANSYS Solver. However, there are certain circumstances when you may want to launch them separately. In particular, you may want to run the two solvers on different machines. In this case, you can use the CFX-Solver Manager to start each solver separately. The procedure is as follows:

1. Ensure that the ANSYS input file and CFX-Solver input file for the run are present on the machine on which you want to run ANSYS Solver. Start CFX-Solver Manager on this machine.
2. Start the ANSYS run with **MFX Run Mode** set to `Start ANSYS only`.
3. Ensure that the CFX-Solver input file for the run is present on the machine on which you want to run CFX-Solver. Start CFX-Solver Manager on this machine.
4. Start the CFX run with **MFX Run Mode** set to `Start CFX only`. You will need to supply a host name and port number which tell CFX-Solver how to communicate with ANSYS solver which is already running. **Host Name** is the machine on which the ANSYS Solver was started. **Port Number** is determined by the ANSYS Solver when it starts, and can be read from the file `<jobname>.port` in the ANSYS working directory once the ANSYS Solver has started. If the CFX-Solver Manager was used to start the ANSYS Solver, then the port number appears in the CCL under `EXECUTION CONTROL/SOLVER STEP CONTROL/PROCESS COUPLING/Host Port`.

Note that this procedure effectively sets up two separate runs as far as the CFX-Solver Manager is concerned; each can be monitored as a separate run.

If you do not have CFX-Solver Manager installed on the machine where ANSYS Solver is going to run, you can start the ANSYS Solver from the ANSYS Launcher instead. In this case, the appropriate host name and port number are displayed in a dialog box from the ANSYS Launcher as the argument `-cplg-host <Port Number>@<Host Name>`.

Process ANSYS Input File Only

The processing of an ANSYS input file is a procedure which takes a solid physics setup in the form of an ANSYS input file and a multi-field setup from a CFX-Solver input file, and combines them into a new ANSYS input file which contains both the solid physics setup and the multi-field setup. The new ANSYS input file has a name of the form `<jobname>.mf` and can be used to start the ANSYS Solver as part of an ANSYS Multi-field run.

Usually the processing takes place as part of launching the ANSYS Solver from the CFX-Solver Manager. However, you may want to do it as a pre-processing step if, for some reason, you want to modify or check the new ANSYS input file before use, or if you want to use it outside of the CFX-Solver Manager. This is also a necessary part of restarting an ANSYS Multi-field run with a change to the solid physics. For details, see Restart Procedure for ANSYS Multi-field Runs (p. 22). The procedure for a non-restart run is as follows:

1. Ensure that the ANSYS input file and CFX-Solver input file for the run are present and start the CFX-Solver Manager.
2. Perform the processing by defining a new run, setting **MFX Run Mode** to `Process Input File only`.
3. Use the new ANSYS input file `<jobname>.mf` in the ANSYS working directory as desired.

If you subsequently want to use this file to start the ANSYS Solver, then click the **MultiField** tab of the **Define Run** dialog box, specify the new file as the ANSYS input file and uncheck **Process ANSYS Input File**.

The processing step is described more fully in Processing the ANSYS Input File (p. 21).

Directory Structure

For a normal CFX run (not an ANSYS Multi-field run), you must specify a working directory in the **Define Run** dialog box. This defaults to the directory from which the CFX-Solver Manager was started.

Once the run is started, a run name is generated. This is the name of the CFX-Solver input file with the extension removed and a three-digit number added. The number is usually 001 if this is the first time CFX-Solver Manager is used for this CFX-Solver input file. In general, it is the lowest number that prevents files from previous runs being overwritten.

Whilst the run is in progress, a CFX working directory with the name `<jobname>.dir` will exist within the CFX working directory, which contains the temporary files for CFX-Solver. When the run has finished, you will find the files `<run_name>.res` and `<run_name>.out` in the CFX working directory, and in certain circumstances (generally when transient results files or backup files are present) a directory `<run_name>` will also be present.

This directory structure is unchanged for the CFX part of an ANSYS Multi-field run. However, a new directory `<run_name>.ansys` is also created in the CFX working directory. This directory is used as the ANSYS working directory by the ANSYS solver, and all files generated by ANSYS will be placed in this directory, including the ANSYS results file.

Almost all files produced by ANSYS have a name of the form `<jobname>.*` (the exception is the ANSYS GST file, which will always be called `ANSYS.gst` when a run is started from the CFX-Solver Manager or ANSYS Launcher). ANSYS Multi-field runs started from the CFX-Solver Manager have a default jobname of `ANSYS`. So, for example, the results file will be `ANSYS.rst` for a case involving structural physics or `ANSYS.rth` for a case involving only thermal physics. Note also that all text output generated by the ANSYS solver (that is, standard error and standard output) is written to the `ANSYS.stdout` file. These files will be in the directory `<run_name>.ansys`. The jobname can be specified explicitly if you start the ANSYS Multi-field run from the command line. For details, see Starting an ANSYS Multi-field Run from the Command Line (p. 20).

Starting an ANSYS Multi-field Run from the Command Line

An ANSYS Multi-field run can be started from the command line using the `cfx5solve` command.

Monitoring a Run in Progress

Each ANSYS Multi-field run launched from the CFX-Solver Manager or the `cfx5solve` command line creates a temporary CFX working directory with the name `<run_name>.dir`, even if the **MFX Run Mode** is set to `Start ANSYS only`. For details, see Directory Structure (p. 20). If a run is already in progress when the CFX-Solver Manager is opened, it can be monitored by selecting this directory as the run directory. The process is just the same as for ordinary CFX runs:

1. Select **File** > **Monitor Run in Progress**.

   The **Select a Run Directory (.dir)** dialog box is displayed.

2. Browse to the directory containing the current run.
3. Select the current run.
4. Click **OK**.

Monitoring a Completed Run

A completed ANSYS Multi-field run can only be monitored in the CFX-Solver Manager if a CFX results file exists for the run. This implies that a run with **MFX Run Mode** set to `Start ANSYS only` or `Process Input File only` cannot be monitored if it has already been completed. If a CFX results file does exist, then a completed run can be monitored in the same way as an ordinary CFX run, as follows:

1. Select **File** > **Monitor Finished Run**.

   The **Monitor Finished Run** dialog box is displayed.

2. Under File type, select the type of files to view. This setting usually corresponds to a CFX results file.
3. Browse to the directory containing the finished run.
4. Select the run to view.

If required, select a different file.

5. Click **OK**.

ANSYS Multi-field Residual Plotting

The extra residual plots for an ANSYS Multi-field run (ANSYS Interface Loads and ANSYS Field Solver plots) are described in Residual Plotting for ANSYS Multi-field Runs (p. 72) and Plot Lines Tab (p. 94).

Processing the ANSYS Input File

The processing of an ANSYS input file is a procedure which takes a solid physics setup in the form of an ANSYS input file and a multi-field setup from a CFX-Solver input file, and combines them into a new ANSYS input file which contains both the solid physics set-up and the multi-field set-up. The new ANSYS input file has a name of the form `<jobname>.mf` and can be used to start the ANSYS Solver as part of an ANSYS Multi-field run.

The following topics are discussed in this section:

- When the ANSYS Input File Needs Processing (p. 21)
- The Processing Step (p. 21)

When the ANSYS Input File Needs Processing

The pre-processing steps for an ANSYS Multi-field run result in the fluid physics set-up (contained in a CFX-Solver input file), the solid physics set-up (contained in an ANSYS input file) and the ANSYS Multi-field settings (contained in either the CFX-Solver input file or the ANSYS input file). When the multi-field settings are only contained in the CFX-Solver input file, then the ANSYS input file must be processed before the run begins to create a new "multi-field" file with the extension `.mf`, which contains both the solid physics from the supplied ANSYS input file and the multi-field settings from the CFX-Solver input file, and this is then used as input for the ANSYS Solver, instead of the supplied ANSYS input file.

A CFX-Solver input file will contain multi-field settings if it was created in `ANSYS MultiField` mode, but not if it was created in `ANSYS MultiField via Prep7` mode.

An ANSYS input file will only contain multi-field settings if these were explicitly added (by hand or in the ANSYS Prep7 user interface); there will be no multi-field settings in an ANSYS input file created by Simulation. A "multi-field" file created from one run can be used as the ANSYS input file for another run, and this will contain multi-field settings.

The Processing Step

When CFX-Pre writes a CFX-Solver input file in `ANSYS MultiField` mode, the CCL in the CFX-Solver input file contains various MFX settings, which are translated to commands for the ANSYS solver when the ANSYS input file is processed. This section contains a few notes on how the MFX settings in the CFX-Solver input file are converted to ANSYS commands.

- Every quantity set in a CFX-Solver input file has units—a timestep has a unit of time, most commonly seconds. For example, timesteps may appear in the CCL as `Timesteps = 0.1 [s]`. You can use any unit of the appropriate dimension to specify any physical quantity, and CFX handles the conversion to a consistent set of units (CFX Solution Units) as part of setting up the solver run. However, ANSYS commands appear without units (for example, `MFDT,0.1,0.1,0.1`) and it is assumed that you have entered the value in units that are consistent with the rest of the ANSYS set-up. When an MFX run is performed, it is a requirement that the units used for the ANSYS set-up match the CFX Solution Units. Therefore, when quantities with units in the CFX CCL are translated to ANSYS commands, the values used are always written in the CFX Solution Unit of the appropriate dimension. For example, if you specify a coupling time duration of `30.0 [s]`, and the CFX Solution Units of time are set to minutes, then the ANSYS command that corresponds to this CCL (MFTI) will be written with the value 0.5 in implicit units of minutes: `MFTI,0.5`.
- The CFX CCL records the name of an ANSYS Interface, not its number, where possible. For details, see Boundary Conditions in ANSYS Multi-field Mode (p. 299) in the ANSYS CFX-Solver Modeling Guide. When the CFX-Solver input file is processed, this is automatically converted to the required interface number, as required by the MFLC command.

- If the CFX CCL contains definitions for backup files or transient results files (of any kind) that are specified with an **Output Frequency** of `Coupling Step Interval` or `Every Coupling Step`, then an `MFOU` command is generated with a corresponding argument, to get ANSYS to write out its results at the same frequency. For example, if **Output Frequency** is set to `Every Coupling Step` then `MFOU,1` is written, and if **Output Frequency** is set to `Coupling Step Interval` with the interval set to every five coupling steps, then `MFOU,5` is generated. If multiple backup and transient results objects are present, then the `MFOU` command which corresponds to the smallest coupling interval is written.
- The processing always adds a `KBC,1` command to the input file, as this is required for any multi-field case.
- The processing always adds the `/GST` command to ensure output for monitoring the run is available, and the `CMWRITE` command to ensure that any named selections or components in the ANSYS input file are available for CFD-Post to use. For details, see ANSYS Files (p. 74) in the ANSYS CFD-Post User's Guide.

Restart Procedure for ANSYS Multi-field Runs

Restarting an ANSYS Multi-field run involves restarting both the ANSYS and CFX Solvers, in such as way that both continue from a consistent point in the ANSYS Multi-field run. A run cannot be restarted with new topology or mesh, but it can be restarted with different physics and/or solver settings.

For restarting the ANSYS part of the run, the key file is the database which contains the settings for the previous run. Various other files are also required for a restart to work correctly, and these will generally be present at the end of a successful run. (If the ANSYS input file for the initial run was prepared manually, it must have a `SAVE` command after the `SOLVE` command for all the required files to be generated.)

The procedure for each type of restart is listed below:

Restart from End of Previous Run

This is the most straight-forward restart and can be performed in almost any circumstance when a run has finished normally (including when the run has been stopped manually). The procedure is as follows:

1. If you stopped the initial run before completion and want to do a restart to finish it off, then the coupling time duration will already be correct in the multi-field set-up in CFX-Pre. However, if the initial run ran to completion and you now want to extend it, you must first set the coupling time duration to have an appropriate end time for the restarted run. This can be done by editing the CCL in the CFX results file from the initial run, or by re-writing the original CFX-Solver input file with the new setting (found on the **Analysis Type** panel).
2. Start the CFX-Solver Manager and open the **Define Run** dialog box. Select the CFX results file from the end of the previous run as the CFX-Solver input file if this contains the correct coupling time duration; otherwise, select the CFX-Solver input file with the correct coupling time duration and use the previous results file as an initial values file.
3. On the **MultiField** tab, enable the **Restart ANSYS Run** toggle, and browse to find the database (`*.db` or `*.rdb`) from the initial run. The database can be found in the directory `<run_name>.ansys` from the previous run if that run was launched by the CFX-Solver Manager or the CFX-Solver script `cfx5solve`.
4. Start the run as normal. Behind the scenes, the CFX-Solver script will copy the required files from the previous ANSYS run into the new ANSYS working directory so that everything should proceed smoothly.

Restart from Intermediate Point During Previous Run

You may want to restart from an intermediate point of a previous run if the initial run was diverging and you want to restart from a point before it was diverging with some modified solver settings, for example.

Assuming that the restart from an intermediate point is available, the procedure is as follows:

1. **Coupling Initial Time** must be set to `Value`, with the value set to the time value that you want to restart from. This can be done by editing the CCL in the "full" CFX backup (`*.bak`) or "full" CFX transient file (`*.trn`) from the initial run, or by re-writing the original CFX-Solver input file with the new setting (found on the **Analysis Type** panel).
2. Start the CFX-Solver Manager and open the **Define Run** dialog box. Select the CFX results file from the end of the previous run as the CFX-Solver input file if this contains the correct coupling initial time; otherwise select the CFX-Solver input file with the correct coupling initial time and use the previous results file as an initial values file.
3. On the **MultiField** tab, enable the **Restart ANSYS Run** toggle, and browse to find the database (`*.rdb`) from the initial run. The database can be found in the directory `<run_name>.ansys` from the previous run if that run was launched by the CFX-Solver Manager or the CFX-Solver script `cfx5solve`.
4. Start the run as normal. Behind the scenes, the CFX-Solver script will copy the required files from the previous ANSYS run into the new ANSYS working directory so that everything should proceed smoothly.

However, restarting from an intermediate point of a previous run is not always possible:

- If the initial run did not complete normally, it depends on the exact circumstances of the failure as to whether it is possible to restart from an intermediate point. The ANSYS Solver requires certain information in order to restart correctly, some of which is only written at the end of a run, and so the ability to restart depends on whether the ANSYS Solver was able to write that information.
- Restarting from an intermediate point is limited to ANSYS runs which contain only mechanical and/or thermal physics. If the ANSYS setup contains additional physics such as electromagnetic physics, then it will not be possible.
- Restarting from an intermediate point relies on the presence of a set of files for a "multiframe restart". For details, see Singleframe and Multiframe Restarts (p. 24). The availability of these files depends on the presence of a command `RESCONTROL` and its setting in the ANSYS input file. In general, ANSYS input files generated from Simulation have the command `RESCONTROL` set so that the set of files required is *not* present. To ensure that the files are present wherever possible, add the command `RESCONTROL,DEFINE,ALL` to the ANSYS input file directly or to a `Commands` object in the relevant `Solution` item in the tree within Simulation. More details can be found in the ANSYS documentation, in "Multiframe Restart". You can check whether the required set of files is present by looking for a database with extension `*.rdb` in the ANSYS working directory from the initial run. If it is present, then the required file set has been written and you should be able to restart from an intermediate point, subject to the conditions regarding how the initial run was terminated noted above.
- Restarting from an intermediate point requires a CFX full backup or full transient file for that time value in addition to having the appropriate ANSYS files for that time value.

Changing CFX Physics or ANSYS Multi-field Settings

Any physics or setup contained in the CFX definition can be modified on a restart, providing that you do not change the geometry, connectivity, mesh or boundary condition names or locations. The changes can be made by editing the CFX results file before using it in a restart, or by writing a modified CFX-Solver input file and using the previous results file as an initial values file.

Changing ANSYS Physics

There is no built-in support for changing the ANSYS physics within Simulation and having it used for restarting an ANSYS Multi-field run. If you want to change the ANSYS setup for a restart, you will have to add ANSYS commands to the ANSYS input file. Commands available for an ANSYS input file are documented in the ANSYS documentation, in *ANSYS Commands Reference*. If you are not already familiar with working with ANSYS commands, then it is recommended that you avoid this type of restart.

The recommended procedure for restarting with different ANSYS physics or solver setup is as follows:

1. Start the CFX-Solver Manager and open the **Define Run** dialog box. Select the appropriate CFX-Solver input file or results file for the restart.
2. On the **MultiField** tab, set **MFX Run Mode** to `Process Input File only`. Also enable the **Restart ANSYS Run** toggle, and browse to find the database (`*.db` or `*.rdb`) from the initial run. The database can be found in the directory `<run_name>.ansys` from the previous run if that run was launched by the CFX-Solver Manager or the CFX-Solver script `cfx5solve`.

3. Start the run as normal. Behind the scenes, the CFX-Solver script will create the ANSYS working directory c and place a file called `ANSYS.mf` inside it.
4. In a text editor, edit the file `ANSYS.mf` to add ANSYS commands to make any changes to the physics that you require.
5. In the CFX-Solver Manager, define a new run. Select the appropriate CFX-Solver input file or results file for the restart.
6. On the **MultiField** tab, set **MFX Run Mode** to `Start ANSYS and CFX`. For **ANSYS Input File**, select the `ANSYS.mf` file that you have just been editing. Toggle off `Process ANSYS Input File`. Ensure that the **Restart ANSYS Run** toggle is still enabled, and browse to find the database (`*.db` or `*.rdb`) from the initial run. The database can be found in the directory `<run_name>.ansys` from the previous run if that run was launched by the CFX-Solver Manager or the CFX-Solver script `cfx5solve`.
7. Start the run. Behind the scenes, the CFX-Solver script will copy the required files from the previous ANSYS run into the new ANSYS working directory so that everything should proceed smoothly.

Restart Limitations

Restarted simulations will give results that are identical to continuous simulations if fields and loads are converged within each multi-field timestep.

Discontinuities in the convergence history upon restarts can be eliminated by running the ANSYS field solver first within each stagger iteration, and executing two or more stagger iterations per multi-field timestep.

Singleframe and Multiframe Restarts

ANSYS has two distinct types of restart: singleframe and multiframe. A full description of what these are is given in the ANSYS documentation, in “Restarting an Analysis”. This section includes some abbreviated information in order to note how restarts from within the CFX-Solver Manager are operated.

- A singleframe restart only allows you to restart a run from where it finished. It is available for any ANSYS physics. The files needed for this type of restart include `<jobname>.db` and various others.
- A multiframe restart allows you to restart from an intermediate point of the initial run (provided that all appropriate files are present), or from the end of the previous run. It is only available for a limited set of ANSYS physics (static or thermal analyses that do not use coupled field elements). The files needed for this type of restart include `<jobname>.db` and various others. Note that multiframe restarts from the end of a previously failed run (due to the solver divergence, for example) are not possible; they restart from an intermediate point instead.

In many cases, either a singleframe or multiframe restart may be performed because both the `<jobname>.db` and `<jobname>.rdb` files exist in the previous run directory. When the cfx5solve script is used, it will automatically copy all of the files required to properly restart an ANSYS Multi-field simulation. It is worth noting, however, that singleframe restarts (that is, by selecting the `<jobname>.db` file) are recommended for simple run continuations since fewer files will be copied from the previous to the new `<jobname>.ansys` run directory.

Chapter 3. CFX-Solver Files

This chapter describes the file types used and generated by CFX-Solver. The CFX-Solver is run using an input file that is usually the CFX-Solver Input file (.def or .mdef) created by CFX-Pre. For most simulations, the CFX-Solver generates text output and CFX-Solver Results (.res, .trn, .bak, .mres) files. Other files are also generated depending on the physical models used in the simulation, and how the simulation is run (e.g. serial or parallel).

Detailed descriptions of these files and how they are used are presented in the following sections:

Files Used by the CFX-Solver

The CFX-Solver input file usually contains all the information that is required by the CFX-Solver to run a CFD simulation. This information includes:

- Physical models and fluid property settings
- Boundary conditions
- Initial conditions
- The mesh
- CFX-Solver parameter settings.

However, there are circumstances when the file specified in the **Solver Input File** option (referred to hereafter as the **Solver Input File**) requires additional solution values to initialize the run. These additional initial solution values are introduced by defining one or more **Initial Values** objects, each of which refers to either a previously created results file or a configuration for which a results file has not yet been created. In all cases, these results files merely supplement the run with solution values that are not available in the input file; simulation specifications in the **Solver Input File** are not overridden.

When starting a run using results from Initial Values objects, the mesh from the **Solver Input File** is used by default and solution values are either copied or interpolated from the initial values mesh(es) onto the **Solver Input File** mesh. Additionally, when initial values files are used, the run history (i.e. monitor and convergence data, simulation time and timestep counters) is continued by default. You may also choose to not continue the run history.

The following tables describe the behavior resulting from different combinations of **Solver Input File** and **Initial Values** objects for single and multi-configuration simulations.

Single Configuration Simulations		
Solver Input File	**Initial Values Specification**	**Description**
CFX-Solver Input file (`.def`)		New simulation (i.e. no run history) starting from iteration or time step # 1.
CFX-Solver Results (`.res`)		Continue simulation (solution values and run history), starting from the iteration or time step that follows last completed in the previous run.
CFX-Solver Input (`.def`)	Initial Values object(s) using the Results File option that references a CFX-Solver Results file (`.res`, `.trn`, `.bak`).	Supplement initial conditions in the **Solver Input File** with solution values contained in the results file referenced by

Single Configuration Simulations		
Solver Input File	**Initial Values Specification**	**Description**
		the Initial Values object(s).

Multi-Configuration Simulations		
Solver Input File	**Configuration-Specific Initial Values Specification**	**Description**
CFX-Solver Input file (`.mdef`)		New simulation (i.e. no run history) with all configuration analyses starting from iteration or time step # 1.
CFX-Solver Input file (`.mdef`)	Initial Values object(s) using the Results File option that references a CFX-Solver Results file (.res, .trn, .bak).	Supplement initial conditions in configuration definitions with those contained in the results file(s) referenced by the Initial Values object(s).
CFX-Solver Input file (`.mdef`)	Initial Values object(s) using the Configuration Results option that references a configuration.	Supplement initial conditions in configuration definitions with solution values contained in the latest results file corresponding to the configuration(s) referenced by the Initial Values object(s).
CFX-Solver Results file (`.mres`)		Continue simulation (solution values and run history), starting from configuration that follows the one last completed in the previous run. The simulation will proceed to complete the last configuration being executed in the previous run if the simulation was stopped prematurely (e.g. via the **cfx5stop** command). Note that his is the only way to continue a multi-configuration simulation.

Note

- For multi-configuration simulations:

  - Global (or simulation) level Initial Values specifications are not valid,
  - A configuration level **Solver Input File** is implied in the multi-configuration setup (i.e. it is not required)
- Setting the **Initial Values Specification** > **Use Mesh From** option to **Solver Input File** will activate the CFX-Interpolator to either copy or interpolate solution values from the mesh in the Initial Values object(s) to the **Solver Input File**. See Using the CFX-Interpolator (p. 86) in ANSYS CFX-Solver Modeling Guide for details.
- Setting the **Initial Values Specification** > **Use Mesh From** option to **Initial Values** will use the mesh from the Initial Values File and de-activate the CFX-Interpolator. . See Using the Mesh from the Initial Values File (p. 84) in ANSYS CFX-Solver Modeling Guide for details.
- Unsetting (i.e. deselecting) the **Initial Values Specification** > **Continue History From** option will reset the run history. and use the Initial Values File to provide a basic initial guess for the new run. See Continuing the History (p. 82) in ANSYS CFX-Solver Modeling Guide for details
- Setting the **Initial Values Specification** > **Continue History From** option will continue the run history from the results file referenced by the specified Initial Values object, and produce the cleanest restart possible from the Initial Values File. The first iteration or timestep executed follows the last one completed in the referenced results file. See Continuing the History (p. 82) in ANSYS CFX-Solver Modeling Guide for details.

Files Generated by the CFX-Solver

The CFX-Solver typically generates two files for each run: the CFX-Solver Output file and the CFX-Solver Results file. These are discussed in the sections:

- CFX-Solver Output File (p. 28)
- CFX-Solver Output File (Transient Runs) (p. 44)
- CFX-Solver Output File (Interpolation Runs) (p. 45)
- CFX-Solver Output File (Parallel Runs) (p. 46)
- CFX-Solver Output File (Mesh Adaption Runs) (p. 49)
- CFX-Solver Output File (Remeshing Runs) (p. 49)
- CFX-Solver Output File (Conjugate Heat Transfer Runs) (p. 50)
- CFX-Solver Output File (GGI Runs) (p. 50)
- CFX-Solver Output File (Combustion Runs) (p. 52)
- CFX-Solver Output File (Particle Runs) (p. 54)
- CFX-Solver Output File (ANSYS Multi-field Runs) (p. 57)
- CFX-Solver Output File (Radiation Runs) (p. 58)
- CFX-Solver Results File (p. 60)

Additional files are generated for the run as follows:

- For runs that involve the use of the Discrete Transfer or Monte Carlo radiation models, an additional file containing radiation data can also be generated. See CFX Radiation File (p. 61).
- For parallel runs of the CFX-Solver, an additional CFX partition file can also be generated. See CFX Partition File (p. 63).
- For ANSYS Multi-field runs, additional files are created. See Additional Files for ANSYS Multi-field Runs (p. 63).

For multi-configuration simulations, the CFX-Solver generates two additional simulation level text output and results files in addition to the output and results files generated per configuration. These files are discussed in the sections:

- CFX Multi-Configuration Output File (p. 64)
- CFX Multi-Configuration Results File (p. 69)

CFX-Solver Output File

The CFX-Solver Output file is a formatted text file generated by the CFX-Solver and contains information about your CFX model setup, the state of the solution during execution of the CFX-Solver, and analysis statistics for the particular run. This is the same information written to the text output window of the CFX-Solver Manager. For details, see Text Output Window (p. 3).

The file is formatted and divided into sections to allow quick and easy interpretation. The sections which are present for any calculation may depend upon which physical models are being used (that is, whether the model is transient or steady-state) and whether the CFX-Solver is being run as several parallel processes or as a single process.

The CFX-Solver will generate an output file with a name based on the CFX-Solver input file. For example, running the CFX-Solver using the input file named `file.def` in a clean working directory will generate an output file named `file_001.out`.

Header

The header is written at the start of every CFX-Solver Output file and contains information regarding the command which started the job. This information is used to check which files were used to start the run.

CFX Command Language for the Run

The CFX Command Language section describes the problem definition, including domain specification, boundary conditions, meshing parameters and solver control.

The section for the command file looks similar to the following:

```
+--------------------------------------------------------------------+
|                                                                    |
|                    CFX Command Language for Run                    |
|                                                                    |
+--------------------------------------------------------------------+

LIBRARY:
  MATERIAL: Water
    Material Description = Water (liquid)
    Material Group = Water Data, Constant Property Liquids
    Option = Pure Substance
    Thermodynamic State = Liquid
    PROPERTIES:
      Option = General Material
      EQUATION OF STATE:
        Density = 997.0 [kg m^-3]
        Molar Mass = 18.02 [kg kmol^-1]
        Option = Value
      END
      SPECIFIC HEAT CAPACITY:
        Option = Value
        Specific Heat Capacity = 4181.7 [J kg^-1 K^-1]
        Specific Heat Type = Constant Pressure
      END
      REFERENCE STATE:
        Option = Specified Point
        Reference Pressure = 1 [atm]
        Reference Specific Enthalpy = 0.0 [J/kg]
        Reference Specific Entropy = 0.0 [J/kg/K]
        Reference Temperature = 25 [C]
      END
      DYNAMIC VISCOSITY:
        Dynamic Viscosity = 8.899E-4 [kg m^-1 s^-1]
        Option = Value
```

```
      END
      THERMAL CONDUCTIVITY:
        Option = Value
        Thermal Conductivity = 0.6069 [W m^-1 K^-1]
      END
      ABSORPTION COEFFICIENT:
        Absorption Coefficient = 1.0 [m^-1]
        Option = Value
      END
      SCATTERING COEFFICIENT:
        Option = Value
        Scattering Coefficient = 0.0 [m^-1]
      END
      REFRACTIVE INDEX:
        Option = Value
        Refractive Index = 1.0 [m m^-1]
      END
      THERMAL EXPANSIVITY:
        Option = Value
        Thermal Expansivity = 2.57E-04 [K^-1]
      END
    END
  END
END
FLOW: Flow Analysis 1
  SOLUTION UNITS:
    Angle Units = [rad]
    Length Units = [m]
    Mass Units = [kg]
    Solid Angle Units = [sr]
    Temperature Units = [K]
    Time Units = [s]
  END
  ANALYSIS TYPE:
    Option = Steady State
    EXTERNAL SOLVER COUPLING:
      Option = None
    END
  END
  DOMAIN: Default Domain
    Coord Frame = Coord 0
    Domain Type = Fluid
    Location = B1.P3
    BOUNDARY: Default Domain Default
      Boundary Type = WALL
      Location = F1.B1.P3,F2.B1.P3,F4.B1.P3,F5.B1.P3,F6.B1.P3,F8.B1.P3
      BOUNDARY CONDITIONS:
        HEAT TRANSFER:
          Option = Adiabatic
        END
        MASS AND MOMENTUM:
          Option = No Slip Wall
        END
        WALL ROUGHNESS:
          Option = Smooth Wall
        END
      END
    END
```

```
BOUNDARY: in1
  Boundary Type = INLET
  Location = in1
  BOUNDARY CONDITIONS:
    FLOW REGIME:
      Option = Subsonic
    END
    HEAT TRANSFER:
      Option = Static Temperature
      Static Temperature = 315 [K]
    END
    MASS AND MOMENTUM:
      Normal Speed = 2 [m s^-1]
      Option = Normal Speed
    END
    TURBULENCE:
      Option = Medium Intensity and Eddy Viscosity Ratio
    END
  END
END
BOUNDARY: in2
  Boundary Type = INLET
  Location = in2
  BOUNDARY CONDITIONS:
    FLOW REGIME:
      Option = Subsonic
    END
    HEAT TRANSFER:
      Option = Static Temperature
      Static Temperature = 285 [K]
    END
    MASS AND MOMENTUM:
      Normal Speed = 2 [m s^-1]
      Option = Normal Speed
    END
    TURBULENCE:
      Option = Medium Intensity and Eddy Viscosity Ratio
    END
  END
END
BOUNDARY: out
  Boundary Type = OUTLET
  Location = out
  BOUNDARY CONDITIONS:
    FLOW REGIME:
      Option = Subsonic
    END
    MASS AND MOMENTUM:
      Option = Average Static Pressure
      Pressure Profile Blend = 0.05
      Relative Pressure = 0 [Pa]
    END
    PRESSURE AVERAGING:
      Option = Average Over Whole Outlet
    END
  END
END
DOMAIN MODELS:
```

```
    BUOYANCY MODEL:
      Option = Non Buoyant
    END
    DOMAIN MOTION:
      Option = Stationary
    END
    MESH DEFORMATION:
      Option = None
    END
    REFERENCE PRESSURE:
      Reference Pressure = 1 [atm]
    END
  END
  FLUID DEFINITION: Water
    Material = Water
    Option = Material Library
    MORPHOLOGY:
      Option = Continuous Fluid
    END
  END
  FLUID MODELS:
    COMBUSTION MODEL:
      Option = None
    END
    HEAT TRANSFER MODEL:
      Option = Thermal Energy
    END
    THERMAL RADIATION MODEL:
      Option = None
    END
    TURBULENCE MODEL:
      Option = k epsilon
    END
    TURBULENT WALL FUNCTIONS:
      Option = Scalable
    END
  END
END
OUTPUT CONTROL:
  RESULTS:
    File Compression Level = Default
    Option = Standard
  END
END
SOLVER CONTROL:
  Turbulence Numerics = First Order
  ADVECTION SCHEME:
    Option = Upwind
  END
  CONVERGENCE CONTROL:
    Maximum Number of Iterations = 100
    Minimum Number of Iterations = 1
    Physical Timescale = 2 [s]
    Timescale Control = Physical Timescale
  END
  CONVERGENCE CRITERIA:
    Residual Target = 1.E-4
    Residual Type = RMS
```

```
    END
    DYNAMIC MODEL CONTROL:
      Global Dynamic Model Control = On
    END
  END
END
COMMAND FILE:
  Version = 12.0.1
  Results Version = 12.0
END
SIMULATION CONTROL:
  EXECUTION CONTROL:
    INTERPOLATOR STEP CONTROL:
      Runtime Priority = Standard
      EXECUTABLE SELECTION:
        Double Precision = Off
      END
    END
    PARALLEL HOST LIBRARY:
      HOST DEFINITION: watjpvandoo
        Host Architecture String = winnt-amd64
        Installation Root = D:\Program Files\ANSYS Inc\v%v\CFX
      END
    END
    PARTITIONER STEP CONTROL:
      Multidomain Option = Independent Partitioning
      Runtime Priority = Standard
      EXECUTABLE SELECTION:
        Use Large Problem Partitioner = Off
      END
      PARTITIONING TYPE:
        MeTiS Type = k-way
        Option = MeTiS
        Partition Size Rule = Automatic
      END
    END
    RUN DEFINITION:
      Run Mode = Full
      Solver Input File = \
        D:\Users\jpvandoo\Documentation\Tutorials\examples\StaticMixer.def
    END
    SOLVER STEP CONTROL:
      Runtime Priority = Standard
      EXECUTABLE SELECTION:
        Double Precision = Off
      END
      PARALLEL ENVIRONMENT:
        Number of Processes = 1
        Start Method = Serial
      END
    END
  END
END
```

CFD Job Information

This section describes the job characteristics in terms of the Run mode (sequential or parallel), the machine on which the job was started, and the time and date of the start of the run.

The section for job information looks similar to the following:

```
+--------------------------------------------------------------------+
|                          Job Information                           |
+--------------------------------------------------------------------+
Run mode:       serial run
Host computer:  pesky
Job started:    Thu Nov 20 15:07:27 2008
```

Memory Allocated for the Run

> **Note**
>
> Allocated storage generally exceeds the required storage. 1 word = 4 bytes, 1 Kword = 1000 words, 1 Kbyte = 1024 bytes

The section for memory usage looks similar to the following:

```
+--------------------------------------------------------------------+
|        Memory Allocated for Run  (Actual usage may be less)        |
+--------------------------------------------------------------------+

 Data Type      Kwords  Words/Node  Words/Elem      Kbytes Bytes/Node

 Real           1014.6     1478.98     2348.56      3963.2    5915.92
 Integer        1206.6     1758.85     2792.99      4713.2    7035.41
 Character      2967.7     4326.09     6869.67      2898.1    4326.09
 Logical          65.0       94.75      150.46       253.9     379.01
 Double            3.1        4.51        7.16        24.2      36.06
```

Mesh Statistics

The mesh statistics summarize domain-specific and global (that is, the combination of all domains):

- mesh quality diagnostics,
- the total number of nodes, elements and boundary faces in the mesh, and
- the area fractions of mesh interfaces that were unmapped

Mesh quality diagnostics include measures of mesh orthogonality, expansion and aspect ratio (see Mesh Issues (p. 321)) For each measure, there are value ranges that are considered good, acceptable, and poor (i.e. may produce accuracy or convergence problems). These ranges are annotated with **OK**, **ok**, and **!**, respectively, in the mesh diagnostics summary. The relevant minimum or maximum value is presented for each measure, plus a summary of the percent of the mesh with values in each of the good, acceptable and poor ranges. Note that these percentages are rounded to the nearest integer value. In the sample output presented below, the worst expansion factor is 37, which is considered poor (i.e. annotated with **!**). While slightly more than 1% of the impeller mesh exhibits similarly poor values, less than 1% of the global mesh is considered poor and '<1' is entered in the summary.

The section for mesh statistics looks similar to the following:

```
+--------------------------------------------------------------------+
|                          Mesh Statistics                           |
+--------------------------------------------------------------------+
| Domain Name          | Orthog. Angle | Exp. Factor   | Aspect Ratio |
+----------------------+---------------+---------------+--------------+
|                      | Minimum [deg] |   Maximum     |    Maximum   |
+----------------------+---------------+---------------+--------------+
| impeller             |       34.7 ok |         37  ! |         13 OK |
| tank                 |       61.4 OK |         11 ok |        303 OK |
| Global               |       34.7 ok |         37  ! |        303 OK |
+----------------------+---------------+---------------+--------------+
```

```
|                      |  %!  %ok  %OK | %!  %ok  %OK | %!  %ok  %OK |
+----------------------+---------------+--------------+--------------+
| impe                 |   0   11   89 |  1   32   67 |  0    0  100 |
| tank                 |   0    0  100 |  0    2   98 |  0    3   97 |
| Global               |   0    3   97 | <1   11   89 |  0    2   98 |
+----------------------+---------------+--------------+--------------+

Domain Name : impeller

    Total Number of Nodes                                =        1710

    Total Number of Elements                             =        7554
        Total Number of Tetrahedrons                     =        7206
        Total Number of Prisms                           =         238
        Total Number of Pyramids                         =         110

    Total Number of Faces                                =        1494

Domain Name : tank

    Total Number of Nodes                                =        4558

    Total Number of Elements                             =        3610
        Total Number of Hexahedrons                      =        3610

    Total Number of Faces                                =        2212

Global Statistics :

    Global Number of Nodes                               =        6268

    Global Number of Elements                            =       11164
        Total Number of Tetrahedrons                     =        7206
        Total Number of Prisms                           =         238
        Total Number of Hexahedrons                      =        3610
        Total Number of Pyramids                         =         110

    Global Number of Faces                               =        3706

Domain Interface Name : ImpellerPeriodic

    Non-overlap area fraction on side 1                  =    0.00E+00
    Non-overlap area fraction on side 2                  =    0.00E+00

Domain Interface Name : TankPeriodic1 TankPeriodic2

    Non-overlap area fraction on side 1                  =    0.00E+00
    Non-overlap area fraction on side 2                  =    0.00E+00
```

Initial Average Scales

These are average scales based on the initial flow field. If the initial velocity field is zero, then the initial average velocity scale will also be zero.

The section for initial average scales looks similar to the following:

```
+--------------------------------------------------------------------+
|                      Average Scale Information                     |
+--------------------------------------------------------------------+
```

```
Domain Name : StaticMixer                                    =  3.2113E+00
Global Length
     Density                                                 =  9.9800E+02
     Dynamic Viscosity                                       =  1.0000E-03
     Velocity                                                =  0.0000E+00
     Thermal Conductivity                                    =  5.9100E-01
     Specific Heat Capacity at Constant Pressure             =  4.1900E+03
     Prandtl Number                                          =  7.0897E+00
```

Checking for Isolated Fluid Regions

For serial runs, the solver checks to see if any fluid domain contains volumetric regions which are isolated pockets. This check cannot be performed for parallel solver runs.

Solved Equations

This section lists the dependent variables solved and the equations to which they relate as well as the estimated physical timestep if calculated automatically.

Equations are given two labels: the individual name and a combined name used for combining residuals together. Residuals for multi-domain problems are combined provided the domains are connected together and have the same domain type (solid or fluid/porous). If there are multiple groups of the same domain type, then the group residual is identified by the name of the first domain in the connected group.

The section for solved equations looks similar to the following:

```
+--------------------------------------------------------------------+
|              The Equations Solved in This Calculation              |
+--------------------------------------------------------------------+
Subsystem : Momentum and Mass
  U-Mom
  V-Mom
  W-Mom
  P-Mass
Subsystem : Thermal Radiation
  I-Radiation
Subsystem : Heat Transfer
  H-Energy
Subsystem : Temperature Variance
  T-Variance
Subsystem : TurbKE and Diss.K
  K-TurbKE
  E-Diss.K
Subsystem : Mixture Fraction
  Z-Mean
  Z-Variance
Subsystem : Mass Fractions
  NO-Mass Fraction
```

Convergence History

The convergence history section details the state of the solution as it progresses. Equation residual information at specified locations allows you to monitor the convergence. Convergence difficulties can often be pinpointed to a particular part of the solution (for example, the momentum equation), and/or a particular location.

The tables shown in the convergence history have the following columns:

- Rate

  The rate is defined as seen in Equation 3.1 (p. 36) where R_n is the residual at iteration n, and R_{n-1} is the residual at an earlier iteration. Rates less than 1.0 indicate convergence.

$$\text{Rate} = \frac{R_n}{R_{n-1}} \quad \text{(Eq. 3.1)}$$

- RMS Res

  The value of the root mean square normalized residual taken over the whole domain.

- Max Res

  The value of the maximum normalized residual in the domain.

- Linear Solution

  The three columns in this section refer to the performance of the linear (inner) solvers. The first column is the average number of iterations the linear solvers attempted to obtain the specified linear equation convergence criteria (within a specified number of iterations). The second column gives the normalized residuals for the solutions to the linear equation. The last column can have one of four entries:

  - `*` indicates that there was a numerical floating point exception and this resulted in the failure of the linear solvers.
  - `F` indicates that the linear solvers did not reduce the residuals (that is, the solution was diverging), but the linear solvers may carry on if the divergence is not catastrophic.
  - `ok` indicates that the residuals were reduced, but that the degree of reduction did not meet the specified criteria.
  - `OK` indicates that the specified convergence criteria for the reduction of residuals was achieved.

After the convergence criteria has been achieved, or the specified number of timesteps has been reached, CFX-Solver appends additional information, calculated from the solution, to the &cfxsolvout-tl; file.

The convergence history for a steady state analysis looks similar to the following:

```
+--------------------------------------------------------------------+
|                        Convergence History                         |
+--------------------------------------------------------------------+
======================================================================
OUTER LOOP ITERATION =    1                     CPU SECONDS = 2.68E+00
----------------------------------------------------------------------
|       Equation       | Rate | RMS Res | Max Res |  Linear Solution |
+----------------------+------+---------+---------+------------------+
| U - Mom              | 0.00 | 1.5E-10 | 5.4E-09 |       1.5E+10  ok|
| V - Mom              | 0.00 | 1.6E-04 | 3.2E-03 |       6.4E+01  ok|
| W - Mom              | 0.00 | 2.5E-10 | 6.4E-09 |       1.1E+10  ok|
| P - Mass             | 0.00 | 2.2E-03 | 3.0E-02 | 12.0  9.5E-02  OK|
+----------------------+------+---------+---------+------------------+
| H-Energy             | 0.00 | 3.6E-03 | 3.6E-02 |  5.4  8.0E-03  OK|
+----------------------+------+---------+---------+------------------+
======================================================================
OUTER LOOP ITERATION =    2                     CPU SECONDS = 1.24E+01
----------------------------------------------------------------------
|       Equation       | Rate | RMS Res | Max Res |  Linear Solution |
+----------------------+------+---------+---------+------------------+
| U - Mom              |99.99 | 5.2E-03 | 7.5E-02 |       4.8E-02  OK|
| V - Mom              |99.76 | 1.6E-02 | 1.6E-01 |       1.4E-02  OK|
| W - Mom              |99.99 | 8.4E-03 | 1.2E-01 |       7.9E-02  OK|
| P - Mass             | 4.26 | 9.3E-03 | 1.3E-01 |  8.3  8.4E-02  OK|
+----------------------+------+---------+---------+------------------+
| H-Energy             | 0.35 | 1.3E-03 | 8.8E-03 |  9.4  2.9E-03  OK|
+----------------------+------+---------+---------+------------------+
.................
.................
======================================================================
OUTER LOOP ITERATION =   29                     CPU SECONDS = 2.44E+02
```

```
------------------------------------------------------------------------
|       Equation       | Rate | RMS Res | Max Res |  Linear Solution |
+----------------------+------+---------+---------+------------------+
| U - Mom              | 0.86 | 7.7E-05 | 3.1E-04 |       5.1E-02  OK|
| V - Mom              | 0.86 | 9.1E-05 | 3.7E-04 |       4.8E-02  OK|
| W - Mom              | 0.86 | 1.9E-05 | 1.6E-04 |       5.0E-02  OK|
| P - Mass             | 0.87 | 3.7E-05 | 1.7E-04 |  8.3  3.0E-02  OK|
+----------------------+------+---------+---------+------------------+
| H-Energy             | 0.86 | 5.7E-06 | 5.5E-05 |  9.5  9.8E-03  OK|
+----------------------+------+---------+---------+------------------+
CFD Solver finished: Wed Oct 25 16:01:48 2000
CFD Solver wall clock seconds: 1.0000E+00
```

If a steady state analysis is continued, then the outer loop iterations and CPU seconds for the current run are enclosed in parenthesis, as shown below. Values not enclosed in parenthesis are the totals for the overall analysis.

```
========================================================================
OUTER LOOP ITERATION =   30 (    1) CPU SECONDS = 2.48E+02 ( 3.38E+00)
------------------------------------------------------------------------
|       Equation       | Rate | RMS Res | Max Res |  Linear Solution |
+----------------------+------+---------+---------+------------------+
| U - Mom              | 0.00 | 1.5E-04 | 1.2E-03 |       4.6E-02  OK|
| V - Mom              | 0.00 | 2.1E-04 | 1.2E-03 |       3.8E-02  OK|
| W - Mom              | 0.00 | 1.5E-04 | 2.1E-03 |       1.3E-02  OK|
| P - Mass             | 0.00 | 3.2E-05 | 1.5E-04 |  8.3  3.4E-02  OK|
+----------------------+------+---------+---------+------------------+
| H-Energy             | 0.00 | 7.7E-06 | 8.5E-05 |  9.5  9.1E-03  OK|
+----------------------+------+---------+---------+------------------+
```

Computed Model Constants

If the Zero Equation model is used to model turbulence, the overall turbulence viscosity is provided.

The section for computed model constants looks similar to the following:

```
+--------------------------------------------------------------------+
|                      Computed Model Constants                      |
+--------------------------------------------------------------------+
Turbulence viscosity for Turbulence Model 1   = 3.3667E+00
```

Termination and Interrupt Condition Summary

After executing each coefficient iteration and time step (or outer iteration), the solver evaluates all internal termination conditions and user-defined interrupt control conditions. After a termination or interrupt, any of these conditions that are true are reported in the CFX-Solver Output file as outlined below.

```
 ====================================================================
              Termination and Interrupt Condition Summary
 ====================================================================
CFD Solver: <internal termination condition description>
User: <interrupt condition object name>
```

Global Conservation Statistics

Global conservation statistics are generated for all transport equations with the exception of turbulence equations that have special wall treatment. These are good checks for the convergence of a solution. Small values of global imbalance indicate that conservation has essentially been achieved. The percentage imbalance of a quantity is calculated as follows:

$$\%\ \text{imbalance} = \frac{\text{Domain Imbalance}}{\text{Maximum Over All Domains}}$$

where "Maximum Over All Domains" is the maximum absolute value, over all boundaries and sources, in all domains. For domain sources, with the exception of user defined source terms, the positive and negative contributions are balanced separately.

For details, see Monitoring and Obtaining Convergence (p. 336) in the ANSYS CFX-Solver Modeling Guide.

The section for global conservation statistics looks similar to the following:

```
================================================================================
             Boundary Flow and Total Source Term Summary
================================================================================

+--------------------------------------------------------------------------+
|                                U-Mom                                     |
+--------------------------------------------------------------------------+
Boundary         : CatConv Default                          -9.4754E-01
Boundary         : Inlet                                     1.6971E+00
Boundary         : InletSide Side 1                          1.3262E-05
Boundary         : Outlet                                   -1.0523E+00
Boundary         : OutletSide Side 1                        -1.7085E-07
Sub-Domain       : catalyst                                  2.6129E-01
Neg Accumulation : CatConv                                   4.1436E-02
Domain Interface : InletSide (Side 1)                       -1.1166E-02
Domain Interface : InletSide (Side 2)                        1.1166E-02
Domain Interface : OutletSide (Side 1)                      -1.4308E-05
Domain Interface : OutletSide (Side 2)                       1.4308E-05
                                                            -----------
Domain Imbalance :                                          -3.6059E-05

Domain Imbalance, in %:                                       -0.0002 %

+--------------------------------------------------------------------------+
|                                V-Mom                                     |
+--------------------------------------------------------------------------+
Boundary         : CatConv Default                          -3.3355E-02
Boundary         : Inlet                                     1.1009E-07
Boundary         : InletSide Side 1                          7.2564E-06
Boundary         : Outlet                                   -4.7561E-04
Boundary         : OutletSide Side 1                         1.9734E-08
Sub-Domain       : catalyst                                  3.4056E-02
Neg Accumulation : CatConv                                  -2.4319E-04
Domain Interface : InletSide (Side 1)                       -4.7260E-04
Domain Interface : InletSide (Side 2)                        4.7262E-04
Domain Interface : OutletSide (Side 1)                       7.5672E-06
Domain Interface : OutletSide (Side 2)                      -7.5672E-06
                                                            -----------
Domain Imbalance :                                          -1.0275E-05

Domain Imbalance, in %:                                        0.0000 %

+--------------------------------------------------------------------------+
|                                W-Mom                                     |
+--------------------------------------------------------------------------+
Boundary         : CatConv Default                          -1.1450E+01
Boundary         : Inlet                                    -1.6971E+00
Boundary         : InletSide Side 1                          1.3180E-01
```

```
Boundary         : Outlet                                  1.0628E+00
Boundary         : OutletSide Side 1                      -6.8369E-02
Sub-Domain       : catalyst                                1.2137E+01
Neg Accumulation : CatConv                                -1.8195E-01
Domain Interface : InletSide (Side 1)                      2.2715E+01
Domain Interface : InletSide (Side 2)                     -2.2592E+01
Domain Interface : OutletSide (Side 1)                    -1.0548E+01
Domain Interface : OutletSide (Side 2)                     1.0493E+01
                                                          -----------
Domain Imbalance :                                         1.4353E-03

Domain Imbalance, in %:                                      0.0063 %

+--------------------------------------------------------------------+
|                              P-Mass                                |
+--------------------------------------------------------------------+
Boundary         : Inlet                                   2.8382E-02
Boundary         : Outlet                                 -5.6938E-02
Neg Accumulation : CatConv                                 2.8576E-02
Domain Interface : InletSide (Side 1)                     -3.0665E-02
Domain Interface : InletSide (Side 2)                      3.0665E-02
Domain Interface : OutletSide (Side 1)                     5.6543E-02
Domain Interface : OutletSide (Side 2)                    -5.6543E-02
                                                          -----------
Domain Imbalance :                                         2.0828E-05

Domain Imbalance, in %:                                      0.0366 %

+--------------------------------------------------------------------+
|                           I-Radiation                              |
+--------------------------------------------------------------------+
Boundary         : CatConv Default                        -1.4392E+01
Boundary         : Inlet                                   1.9442E+01
Boundary         : InletSide Side 1                        3.8307E-01
Boundary         : Outlet                                 -6.1793E+00
Boundary         : OutletSide Side 1                      -1.7803E-01
Domain Src (Neg) : CatConv                                -2.9646E-01
Domain Src (Pos) : CatConv                                 3.0447E-01
Domain Interface : InletSide (Side 1)                     -4.6702E+02
Domain Interface : InletSide (Side 2)                      4.6702E+02
Domain Interface : OutletSide (Side 1)                    -7.4487E+01
Domain Interface : OutletSide (Side 2)                     7.4490E+01
                                                          -----------
Domain Imbalance :                                        -9.0640E-01

Domain Imbalance, in %:                                      -0.1941 %

+--------------------------------------------------------------------+
|                             H-Energy                               |
+--------------------------------------------------------------------+
Boundary         : CatConv Default                         1.4284E+01
Boundary         : Inlet                                   8.6049E+03
Boundary         : InletSide Side 1                       -3.8344E-01
Boundary         : Outlet                                 -2.3960E+02
Boundary         : OutletSide Side 1                       1.7809E-01
Domain Src (Neg) : CatConv                                -3.0461E-01
Domain Src (Pos) : CatConv                                 2.9616E-01
Neg Accumulation : CatConv                                -8.3717E+03
```

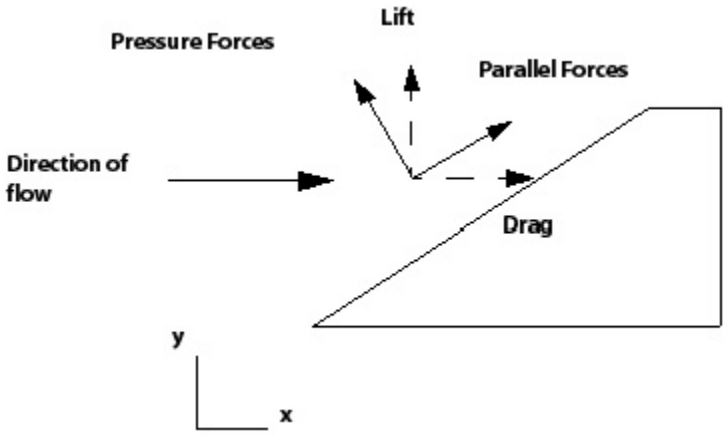


```
 Domain Interface : InletSide (Side 1)                    -7.6620E+03
 Domain Interface : InletSide (Side 2)                     7.6620E+03
 Domain Interface : OutletSide (Side 1)                    1.3624E+02
 Domain Interface : OutletSide (Side 2)                   -1.3624E+02
                                                          -----------
 Domain Imbalance :                                        7.6699E+00

 Domain Imbalance, in %:                                     0.0891 %
```

Calculated Wall Forces and Moments

The CFX-Solver calculates the pressure and viscous components of forces on all boundaries specified as walls. The drag force on any wall can be calculated from these values as follows:



Lift is the net force on the body in the direction perpendicular to the direction of flow. In the above diagram, the lift is the sum of the forces on the wall in the vertical direction, that is, the sum of the pressure force and the viscous force components in the *y* direction.

Drag is the net force on the body in the direction of the flow. In the above diagram, the drag is the sum of the forces on the wall in the horizontal direction, that is, the sum of the pressure force and the viscous force components in the *x* direction.

It is apparent from this that viscous force is not a pure shear force since it also has a small component in the normal direction, arising in part from a normal component in the laminar flow shear stress.

The pressure and viscous moments are related to the pressure and viscous forces calculated at the wall. The pressure moment is the vector product of the pressure force vector F_n and the position vector r. The viscous moment is the vector product of the viscous force vectorF_t and the position vector r. As an example, review Equation 3.2 (p. 40) where M_n and M_t are the pressure and viscous moments respectively.

$$\boldsymbol{M}_n = \vec{\boldsymbol{r}} \times \vec{\boldsymbol{F}}_n$$
$$\boldsymbol{M}_t = \vec{\boldsymbol{r}} \times \vec{\boldsymbol{F}}_t \qquad \text{(Eq. 3.2)}$$

These are summed over all the surface elements in the wall.

It is important to note that forces are evaluated in the local reference frame and that they do not include reference pressure effects. The pressure force is calculated as the integral of the relative pressure over the wall area and not as the integral of the sum of the reference and relative pressures. You can include reference pressure effects in the force calculation by setting the expert parameter `include pref in forces = t`.

It is also important to note that for rotating domains in a transient run, forces are evaluated in the reference frame fixed to the initial domain orientation. These quantities are not influenced by any rotation that might occur during a transient run or when a rotational offset is specified. However, results for rotating domains in a transient run may

be in the rotated position (depending on the setting of **Options** in CFD-Post) when they are subsequently loaded into CFD-Post for post-processing.

The sections for calculated wall forces and moments look similar to the following:

```
================================================================================
                     Wall Force and Moment Summary
================================================================================

Notes:
1. Pressure integrals exclude the reference pressure.  To include
   it, set the expert parameter 'include pref in forces = t'.


+------------------------------------------------------------------------+
|                        Pressure Force On Walls                         |
+------------------------------------------------------------------------+
                                    X-Comp.      Y-Comp.      Z-Comp.

Domain Group: Bottom Box

Copy of Walls                      7.5841E+01   8.4350E+01  -5.2096E+02
Top Box Default                   -5.4841E+01   0.0000E+00   0.0000E+00
Walls                              4.8468E+01   1.2755E+02   5.0230E+02
                                   ----------   ----------   ----------
Domain Group Totals :              6.9468E+01   2.1190E+02  -1.8659E+01


+------------------------------------------------------------------------+
|                        Viscous Force On Walls                          |
+------------------------------------------------------------------------+
                                    X-Comp.      Y-Comp.      Z-Comp.

Domain Group: Bottom Box

Copy of Walls                     -3.4280E-04  -1.8638E-03  -5.7310E-04
Top Box Default                   -4.7982E-05   5.5391E-06  -2.2129E-05
Walls                              2.9180E-04  -4.3111E-05  -6.2137E-04
                                   ----------   ----------   ----------
Domain Group Totals :             -9.8980E-05  -1.9014E-03  -1.2166E-03
```

Maximum Residual Statistics

The location and occurrence of maximum and peak residuals can help to pinpoint convergence difficulties and help to indicate the state of convergence of your solution.

The sections for maximum residual statistics look similar to the following:

```
+--------------------------------------------------------------------+
|                  Locations of Maximum Residuals                    |
+--------------------------------------------------------------------+
|       Equation       |  Node # |     X     |     Y     |     Z     |
+--------------------------------------------------------------------+
| U - Mom              |     728 | 6.168E-01 | 5.387E-02 |-6.187E-01 |
| V - Mom              |      85 | 6.136E-01 | 1.902E+00 | 6.846E-01 |
| W - Mom              |     483 |-6.772E-01 | 2.826E-01 |-8.441E-01 |
| P - Mass             |     413 | 7.552E-01 |-3.366E-01 |-7.821E-01 |
| H-Energy             |    2083 | 6.121E-01 | 1.640E+00 | 1.178E+00 |
+--------------------------------------------------------------------+
```



```
|                      Peak Values of Residuals                      |
+--------------------------------------------------------------------+
|       Equation       |   Loop #  |  Peak Residual | Final Residual |
+--------------------------------------------------------------------+
| U - Mom              |        2  |   5.22793E-03  |   7.67906E-05  |
| V - Mom              |        2  |   1.55203E-02  |   9.05021E-05  |
| W - Mom              |        2  |   8.44563E-03  |   1.87603E-05  |
| P - Mass             |        2  |   9.33986E-03  |   3.72930E-05  |
| H-Energy             |        1  |   3.58668E-03  |   5.74081E-06  |
+--------------------------------------------------------------------+
```

False Transient Information

This is only applicable to steady-state simulations (serial and parallel). The information is equation based, that is there is one line per equation solved. For each equation, the type of timestepping used is displayed as `Auto`, `Physical` or `Local`.

Both `Auto` and `Physical` run as false transients. This means that although the simulation is steady state, a transient term with an associated timestep is used to relax the equations during convergence. In this case, the total elapsed pseudo-time is also printed.

The section for false transient information looks similar to the following:

```
+--------------------------------------------------------------------+
|                  False Transient Information                       |
+--------------------------------------------------------------------+
|       Equation       |       Type        |   Elapsed Pseudo-Time   |
+--------------------------------------------------------------------+
| U - Mom              | Physical          |      5.80000E+01        |
| V - Mom              | Physical          |      5.80000E+01        |
| W - Mom              | Physical          |      5.80000E+01        |
| P - Mass             | Physical          |      5.80000E+01        |
| H-Energy             | Physical          |      5.80000E+01        |
+--------------------------------------------------------------------+
```

Final Average Scales

These are average scales for the final flow field.

The section for final average scales looks similar to the following:

```
+--------------------------------------------------------------------+
|                    Average Scale Information                       |
+--------------------------------------------------------------------+
Domain Name : StaticMixer
Global Length                                          =  3.2113E+00
 Density                                             =  9.9800E+02
 Dynamic Viscosity                                   =  1.0000E-03
 Velocity                                            =  1.4534E+00
 Advection Time                                      =  2.2095E+00
 Reynolds Number                                     =  4.6581E+06
 Thermal Conductivity                                =  5.9100E-01
 Specific Heat Capacity at Constant Pressure         =  4.1900E+03
 Prandtl Number                                      =  7.0897E+00
 Temperature Range                                   =  3.0008E+01
```

Variable Range Information

These are the maximum and minimum values for each variable in the flow field.

The section for variable range information looks similar to the following:

```
+--------------------------------------------------------------------+
|                    Variable Range Information                      |
+--------------------------------------------------------------------+
Domain Name : StaticMixer
+--------------------------------------------------------------------+
|      Variable Name                          |    min    |    max    |
+--------------------------------------------------------------------+
| Velocity u                                  | -1.65E+00 |  1.61E+00 |
| Velocity v                                  | -2.26E+00 |  2.25E+00 |
| Velocity w                                  | -4.13E+00 |  2.58E-01 |
| Pressure                                    | -6.71E+02 |  1.38E+04 |
| Density                                     |  9.98E+02 |  9.98E+02 |
| Dynamic Viscosity                           |  1.00E-03 |  1.00E-03 |
| Specific Heat Capacity at Constant Pressure|  4.19E+03 |  4.19E+03 |
| Thermal Conductivity                        |  5.91E-01 |  5.91E-01 |
| Thermal Expansivity                         |  2.10E-04 |  2.10E-04 |
| Eddy Viscosity                              |  1.89E+01 |  1.89E+01 |
| Temperature                                 |  2.85E+02 |  3.15E+02 |
| Static Enthalpy                             |  1.19E+06 |  1.32E+06 |
+--------------------------------------------------------------------+
```

CPU Requirements

The section for CPU requirements looks similar to the following:

```
+--------------------------------------------------------------------+
 |           CPU Requirements of Numerical Solution - Total           |
 +--------------------------------------------------------------------+

 Subsystem Name                  Discretization       Linear Solution
                                (secs.   %total)     (secs.   %total)
 ---------------------------------------------------------------------
 Momentum and Mass              2.50E-01  10.5 %     7.81E-02   3.3 %
                                -------- -------     --------  ------
 Subsystem Summary              2.50E-01  10.5 %     7.81E-02   3.3 %

 Variable Updates               1.25E-01    5.3 %
 File Reading                   1.41E-01    5.9 %
 File Writing                   6.25E-02    2.6 %
 Miscellaneous                  1.72E+00   72.4 %
                                --------
 Total                          2.38E+00
```

Job Information

The section for job information looks similar to the following:

```
 +--------------------------------------------------------------------+
 |                          Job Information                           |
 +--------------------------------------------------------------------+
 Host computer: machinename
 Job finished:   Fri Feb 10 15:14:32 2006
 Total CPU time: 9.524E+01 seconds
             or: (          0:         0:         1:    35.240 )
                 (       Days:     Hours:   Minutes:   Seconds )
 Total wall clock time: 2.720E+02 seconds
             or: (          0:         0:         1:    42.000 )
```

```
                  (       Days:     Hours:   Minutes:   Seconds )
End of solution stage.
```

CFX-Solver Output File (Transient Runs)

For transient runs, the CFX-Solver outputs convergence information for each coefficient iteration.

At the completion of each timestep iteration, the following information is also written to the CFX-Solver Output file:

- Global conservation statistics
- Calculated wall forces and moments
- Maximum residual statistics
- Average scale information

Convergence History

The convergence history for a transient analysis looks similar to the following:

```
================================================================================
 TIME STEP =   2   SIMULATION TIME = 5.00E-01    CPU SECONDS = 2.14E+01
 ------------------------------------------------------------------------------
 COEFFICIENT LOOP ITERATION =   1
 ------------------------------------------------------------------------------
 |       Equation       | Rate | RMS Res | Max Res |  Linear Solution |
 +----------------------+------+---------+---------+------------------+
 | U - Mom              |99.99 | 9.6E-01 | 7.7E+00 |       5.8E-04  OK|
 | V - Mom              |99.99 | 2.2E-01 | 2.9E+00 |       2.7E-03  OK|
 | W - Mom              |99.99 | 4.2E-01 | 4.4E+00 |       5.4E-04  OK|
 | P - Mass             |  .12 | 3.0E-03 | 4.0E-02 | 12.0  4.6E-02  OK|
 +----------------------+------+---------+---------+------------------+
 | Smoke                |  .89 | 6.4E-02 | 7.3E-01 |  5.4  5.4E-05  OK|
 +----------------------+------+---------+---------+------------------+
 | K-TurbKE             |  .53 | 2.1E-01 | 5.0E-01 |  5.4  3.9E-05  OK|
 +----------------------+------+---------+---------+------------------+
 | E-Diss.K             |  .73 | 5.2E-01 | 9.5E-01 |  5.4  2.4E-05  OK|
 +----------------------+------+---------+---------+------------------+
 ------------------------------------------------------------------------------
 COEFFICIENT LOOP ITERATION =   2                CPU SECONDS = 2.83E+01
 ------------------------------------------------------------------------------
 |       Equation       | Rate | RMS Res | Max Res |  Linear Solution |
 +----------------------+------+---------+---------+------------------+
 | U - Mom              |  .14 | 1.4E-01 | 1.3E+00 |       2.0E-03  OK|
 | V - Mom              |  .16 | 3.5E-02 | 4.9E-01 |       8.3E-03  OK|
 | W - Mom              |  .14 | 5.8E-02 | 1.0E+00 |       1.6E-03  OK|
 | P - Mass             |  .65 | 1.9E-03 | 3.7E-02 |  8.3  5.3E-02  OK|
 +----------------------+------+---------+---------+------------------+
 | Smoke                |  .29 | 1.9E-02 | 2.6E-01 |  5.4  5.6E-05  OK|
 +----------------------+------+---------+---------+------------------+
 | K-TurbKE             |  .17 | 3.6E-02 | 2.2E-01 |  5.4  8.2E-05  OK|
 +----------------------+------+---------+---------+------------------+
 | E-Diss.K             |  .16 | 8.3E-02 | 2.4E-01 |  5.4  6.4E-05  OK|
 +----------------------+------+---------+---------+------------------+
 ------------------------------------------------------------------------------
 COEFFICIENT LOOP ITERATION =   3                CPU SECONDS = 3.50E+01
 ------------------------------------------------------------------------------
 |       Equation       | Rate | RMS Res | Max Res |  Linear Solution |
 +----------------------+------+---------+---------+------------------+
 | U - Mom              |  .24 | 3.3E-02 | 2.8E-01 |       3.8E-03  OK|
```

```
| V - Mom              |  .23 | 8.0E-03 | 1.1E-01 |       1.4E-02  OK|
| W - Mom              |  .25 | 1.4E-02 | 2.5E-01 |       3.3E-03  OK|
| P - Mass             |  .68 | 1.3E-03 | 2.0E-02 |  8.3  3.6E-02  OK|
+----------------------+------+---------+---------+------------------+
| Smoke                |  .36 | 6.8E-03 | 1.1E-01 |  5.4  6.1E-05  OK|
+----------------------+------+---------+---------+------------------+
| K-TurbKE             |  .38 | 1.4E-02 | 1.2E-01 |  5.4  8.7E-05  OK|
+----------------------+------+---------+---------+------------------+
| E-Diss.K             |  .39 | 3.3E-02 | 1.2E-01 |  5.4  8.4E-05  OK|
+----------------------+------+---------+---------+------------------+
```

If a transient analysis is continued, then the time step counter, simulation time, and CPU seconds for the current run are enclosed in parenthesis below the totals for the overall analysis, as shown below.

```
=======================================================================
 TIME STEP =     3 SIMULATION TIME = 7.50E-01    CPU SECONDS = 2.84E+01
 (THIS RUN:      1                   2.50E-01                  7.00+00)
 -----------------------------------------------------------------------
 COEFFICIENT LOOP ITERATION =   1
 -----------------------------------------------------------------------
 |       Equation       | Rate | RMS Res | Max Res |  Linear Solution |
 +----------------------+------+---------+---------+------------------+
 | U - Mom              |99.99 | 9.6E-01 | 7.7E+00 |       5.8E-04  OK|
 | V - Mom              |99.99 | 2.2E-01 | 2.9E+00 |       2.7E-03  OK|
 | W - Mom              |99.99 | 4.2E-01 | 4.4E+00 |       5.4E-04  OK|
 | P - Mass             |  .12 | 3.0E-03 | 4.0E-02 | 12.0  4.6E-02  OK|
 +----------------------+------+---------+---------+------------------+
 | Smoke                |  .89 | 6.4E-02 | 7.3E-01 |  5.4  5.4E-05  OK|
 +----------------------+------+---------+---------+------------------+
 | K-TurbKE             |  .53 | 2.1E-01 | 5.0E-01 |  5.4  3.9E-05  OK|
 +----------------------+------+---------+---------+------------------+
 | E-Diss.K             |  .73 | 5.2E-01 | 9.5E-01 |  5.4  2.4E-05  OK|
 +----------------------+------+---------+---------+------------------+
```

CFX-Solver Output File (Interpolation Runs)

This section outlines the additional information that is written to the CFX-Solver Output file for CFX-Solver jobs that require interpolation.

```
=======================================================================
                 Interpolating Onto Domain "Bottom Box"
 =======================================================================

 Total Number of Nodes in the Target Domain              =          343
 Bounding Box Volume of the Target Mesh                  =  2.70000E+01


 Checking all source domains from the source file:
   Target mesh is different from domain "Top Box".

 Searching for Candidate Source Domains:

 Warning: The target mesh does not intersect with any source meshes
          that have the same domain type and motion.
          Skip the interpolation.

 =======================================================================
                  Interpolating Onto Domain "Top Box"
```

```
================================================================================

Total Number of Nodes in the Target Domain              =          343
Bounding Box Volume of the Target Mesh                  =  2.70000E+01


Checking all source domains from the source file:
  Target mesh is the same as domain "Top Box".

Start direct copying of variables from domain "Top Box".

+--------------------------------------------------------------------+
|                     Variable Range Information                     |
+--------------------------------------------------------------------+
+--------------------------------------------------------------------+
|      Variable Name                          |    min    |    max    |
+--------------------------------------------------------------------+
| Thermal Conductivity                        |  2.61E-02 |  2.61E-02 |
| Courant Number                              |  9.86E-02 |  4.52E+00 |
| Density                                     |  1.18E+00 |  1.18E+00 |
| Static Entropy                              |  0.00E+00 |  0.00E+00 |
| Pressure                                    |  1.59E+01 |  6.01E+01 |
| Specific Heat Capacity at Constant Pressure|  1.00E+03 |  1.00E+03 |
| Specific Heat Capacity at Constant Volume  |  1.00E+03 |  1.00E+03 |
| Temperature                                 |  2.98E+02 |  2.98E+02 |
| Velocity                                    |  2.02E-01 |  8.09E+00 |
| Dynamic Viscosity                           |  1.83E-05 |  1.83E-05 |
+--------------------------------------------------------------------+
```

CFX-Solver Output File (Parallel Runs)

This section outlines the additional information that is written to the CFX-Solver Output file for CFX-Solver jobs submitted in parallel.

Partitioning Information

If the partitioning step is run, partitioning information pertaining to the current job is displayed. This includes how the mesh is divided, and CPU requirements for the partitioning process.

The section for partitioning information looks similar to the following:

```
+--------------------------------------------------------------------+
|                     Vertex Based Partitioning                      |
+--------------------------------------------------------------------+

Partitioning of domain: Bottom Box

- Partitioning tool:       MeTiS multilevel k-way algorithm
- Number of partitions:             2
- Number of graph-nodes:          343
- Number of graph-edges:         1764

+--------------------------------------------------------------------+
|                      Partitioning Information                      |
+--------------------------------------------------------------------+

Partitioning information for domain: Bottom Box

       +-----------------+-----------------------+-----------------+
```

```
        |     Elements     |        Vertices         |     Faces       |
+------+------------------+-------------------------+-----------------+
| Part |   Number     %   |   Number     %   %Ovlp | Number     %    |
+------+------------------+-------------------------+-----------------+
| Full |       216        |       343               |      216        |
+------+------------------+-------------------------+-----------------+
|    1 |       173   61.8 |       317   61.9  36.6  |      175   62.5 |
|    2 |       107   38.2 |       195   38.1  27.2  |      105   37.5 |
+------+------------------+-------------------------+-----------------+
| Sum  |       280  100.0 |       512  100.0  33.0  |      280  100.0 |
+------+------------------+-------------------------+-----------------+

+---------------------------------------------------------------------+
|                 Partitioning CPU-Time Requirements                  |
+---------------------------------------------------------------------+

- Preparations                                          0.000E+00 seconds
- Low-level mesh partitioning                          0.000E+00 seconds
- Gather zone interface information                    0.000E+00 seconds
- Global partitioning information                      0.000E+00 seconds
- Element and face partitioning information            0.000E+00 seconds
- Vertex partitioning information                      0.000E+00 seconds
- Partitioning information compression                 0.000E+00 seconds
- Summed CPU-time for mesh partitioning                0.000E+00 seconds
```

Job Information

The section for job information includes master and slave partition process details, and looks similar to the following:

```
+---------------------------------------------------------------------+
 |                          Job Information                            |
 +---------------------------------------------------------------------+

 Run mode:       parallel run (MPI)

 Host computer:    Speedy
 Par. Process:     Master running on mesh partition:       1
 Solver Build:     Wed Nov 19 04:07:01 EST 2008
 Attributes:       double-int32-64bit-novc8-noifort-novc6-optimised-su...
 Job started:      Wed Nov 19 15:39:39 2008

 Host computer:    Speedier
 Par. Process:     Slave  running on mesh partition:       2
 Solver Build:     Wed Nov 19 04:07:01 EST 2008
 Attributes:       double-int32-64bit-novc8-noifort-novc6-optimised-su...
 Job started:      Wed Nov 19 15:39:39 2008
```

Host Information

The section for host information looks similar to the following:

```
+---------------------------------------------------------------------+
|                           Host Information                          |
+---------------------------------------------------------------------+
| Name            |  Rel. Speed  | # of Proc. | # of Part  | Stat. Load |
+-----------------+--------------+------------+------------+------------+
| camel           |      1.00000 |          1 |          1 |    100.0 % |
```

```
| giraffe         |      1.00000 |          1 |          1 |    100.0 % |
+-----------------+--------------+------------+------------+------------+
```

Memory Usage Information

The section for memory usage information looks similar to the following:

```
+--------------------------------------------------------------------+
 |        Memory Allocated for Run  (Actual usage may be less)        |
 +--------------------------------------------------------------------+

 Allocated storage in:    Kwords
                          Words/Node
                          Words/Elem
                          Kbytes
                          Bytes/Node

 Partition | Real       | Integer    | Character| Logical  | Double
 ----------+------------+------------+----------+----------+----------
         1 |     2095.0 |     1679.2 |   3417.7 |     65.0 |      8.0
           |    3809.15 |    3053.14 |  6213.99 |   118.18 |    14.55
           |    6914.29 |    5542.00 | 11279.52 |   214.52 |    26.40
           |    16367.4 |     6559.5 |   3337.6 |     63.5 |     62.5
           |   30473.16 |   12212.55 |  6213.99 |   118.18 |   116.36
 ----------+------------+------------+----------+----------+----------
         2 |     1937.7 |     1672.6 |   3417.7 |     65.0 |      8.0
           |    4185.05 |    3612.62 |  7381.63 |   140.39 |    17.28
           |    7598.74 |    6559.39 | 13402.73 |   254.90 |    31.37
           |    15138.1 |     6533.8 |   3337.6 |     63.5 |     62.5
           |   33480.42 |   14450.50 |  7381.63 |   140.39 |   138.23
 ----------+------------+------------+----------+----------+----------
     Total |     4032.7 |     3351.9 |   6835.4 |    130.0 |     16.0
           |    5878.59 |    4886.11 |  9964.13 |   189.50 |    23.32
           |    9334.98 |    7758.96 | 15822.67 |   300.93 |    37.04
           |    31505.5 |    13093.2 |   6675.2 |    127.0 |    125.0
           |   47028.68 |   19544.43 |  9964.13 |   189.50 |   186.59
 ----------+------------+------------+----------+----------+----------
```

Completed Job Information

The section for completed job information looks similar to the following:

```
+--------------------------------------------------------------------+
|                          Job Information                           |
+--------------------------------------------------------------------+
Host computer:  fastmachine1
Par. Process:   Master running on mesh partition:       1
Job finished:   Thu Nov 28 16:44:01 2005
Total CPU time: 9.025E+02 seconds
             or: (          0:         0:        15:     2.517 )
                 (       Days:     Hours:   Minutes:   Seconds )
Host computer:  slowermachine
Par. Process:   Slave  running on mesh partition: 2
Job finished:   Wed Nov 28 16:54:30 2005
Total CPU time: 1.291E+03 seconds
             or: (          0:         0:        21:    31.034 )
                 (       Days:     Hours:   Minutes:   Seconds )
Total wall clock time: 1.146E+03 seconds
            or: (          0:         0:        19:    6.000 )
```

```
                (       Days:     Hours:   Minutes:   Seconds )
--> Master-Partition Nr.  1 reaches final synchronization point!
--> Slave-Partition  Nr.  2 reaches final synchronization point!
End of solution stage.
+--------------------------------------------------------------------+
| The results from this run of the CFX-Solver have been written      |
| to z:\temp\BluntBody_001.res
+--------------------------------------------------------------------+
This run of the CFX-Solver has finished.
```

CFX-Solver Output File (Mesh Adaption Runs)

When a mesh adaption step is complete, the CFX-Solver Manager reports the new meshing information, including the total number of vertices, elements and faces. The CFX-Solver then continues to determine a solution, using the adapted mesh.

The section for mesh refinement looks similar to the following:

```
+--------------------------------------------------------------------+
|                                                                    |
|                          Mesh Refinement                           |
|                                                                    |
+--------------------------------------------------------------------+
Adaption step 2 of 3.
Number of elements initially marked for refinement:         480
Number of elements removed because:
 They already meet the minimum length criteria:               0
 They are in regions not marked for refinement:               0
 They are already in the deepest refinement level:            0
 There are not enough nodes available to refine them:      -425
                                                        ----------
                                                              55
                                                        ----------
Target number of nodes at end of step:                      1512
Number of vertices in the final mesh:                       1999
Number of elements in the final mesh:                       1560
+--------------------------------------------------------------------+
|         Total number of Vertices, Elements, and Faces              |
+--------------------------------------------------------------------+
Domain Name      : nozzle vmi
     Total Number of Nodes                  =       1999
     Total Number of Elements               =       1560
         Total Number of Tetrahedrons       =        146
         Total Number of Hexahedrons        =        818
         Total Number of Pyramids           =        596
     Total number of Faces                  =       1702
```

CFX-Solver Output File (Remeshing Runs)

Each time a run is terminated or interrupted, all remeshing definitions associated with that run are considered for activation. If any definitions are active, then output similar to the following is generated:

```
+--------------------------------------------------------------------+
 |                                                                    |
 |                             Remeshing                              |
 |                                                                    |
 +--------------------------------------------------------------------+
```

```
Remesh Object: Remesh
  Activated by condition: Number of TimeSteps

  Executing remesh command:
     C:\Program Files\ANSYS
     Inc\v110\icemcfd\win64_amd\bin\icemcfd.bat -batch -script
     d:\builds\v120\CFX\etc\Remeshing\icemcfd_Remesh.rpl

CFX Solver Results generated before remeshing have been written to:
  d:\ICEMRemesh\ballvalve1_001\5_oldmesh.res

Text output generated during remeshing has been written to:
  d:\ICEMRemesh\ballvalve1_001\5_remesh.out
```

Each active remesh object is listed, along with the name of the solver interrupt condition(s) that activated it, and the command used to generate the new mesh. Any output generated during remeshing is redirected into a text file, which, along with the CFX-Solver Results file from the previous run, is placed into the final results directory for the run. The locations of these files is listed in the output.

Once the updated meshes have been generated, CFX-Pre loads the CFX-Solver Results file from the previous run, replaces the old meshes with the updated ones, and writes a new CFX-Solver Input file.

A CFX-Solver run is then started with the new CFX-Solver Input file, using the CFX-Solver Results file from the previous run for initial values. Partitioning is also performed for parallel run modes.

CFX-Solver Output File (Conjugate Heat Transfer Runs)

Thermal Energy at Solid Boundary

When a solid domain is included, CFX-Solver Manager reports the thermal energy that flows into and out of the solid, across existing boundaries. If the solid has a source term, the total thermal energy added is included in the output file, along with the global imbalance.

The section for thermal energy at a solid boundary looks similar to the following:

```
+--------------------------------------------------------------------+
|         Boundary T-Energy - 2 Flow and Total Source Term Summary   |
+--------------------------------------------------------------------+
Boundary   : Default Solid               0.0000E+00
Boundary   : Default Fluid Solid2       -3.0742E+06
Subdomain  : heater                      3.0685E+06
                                         -----------
Global T-Energy - 2 Balance:            -5.6725E+03
Global Imbalance, in %:                                    -0.1852 %
```

CFX-Solver Output File (GGI Runs)

Running the CFX Flow Solver for cases that include GGI interfaces is similar to cases without GGI connections. The following differences, however, will be observed:

- At the start of the simulation, all GGI connection conditions are processed. This may require the computational effort of the order of a single iteration or timestep of the flow simulation. If a GGI connection condition is found to contain non-overlapping portions, the percentage of the area of each side that is non-overlapping is reported. This is a useful diagnostic, and should be reviewed to confirm that the expected amount of non-overlapping area has been detected.

  ```
  +--------------------------------------------------------------------+
  |             Total Number of Nodes, Elements, and Faces             |
  +--------------------------------------------------------------------+
  Domain Name : rotor
       Total Number of Nodes                                =       38360
  ```

```
    Total Number of Elements                                =       33202
        Total Number of Hexahedrons                         =       33202
    Total Number of Faces                                   =        9970
 Domain Name : stator
    Total Number of Nodes                                   =       33320
    Total Number of Elements                                =      106660
        Total Number of Tetrahedrons                        =       75265
        Total Number of Prisms                              =       31395
    Total Number of Faces                                   =       17871
 Domain Interface Name : Rotor Periodic
    Non-overlap area fraction on side 1                     =       0.0 %
    Non-overlap area fraction on side 2                     =       0.0 %
 Domain Interface Name : Stator Periodic
    Non-overlap area fraction on side 1                     =       0.0 %
    Non-overlap area fraction on side 2                     =       0.0 %
```

- Residual diagnostics will be reported for each set of connected domains having the same physical type. For example, the mass and momentum equation residuals will be reported independently for each flow passage.

```
=======================================================================
OUTER LOOP ITERATION =   35                      CPU SECONDS = 4.64E+03
-----------------------------------------------------------------------
|       Equation       | Rate | RMS Res | Max Res |  Linear Solution |
+----------------------+------+---------+---------+------------------+
| U-Mom-rotor          | 0.92 | 2.2E-05 | 3.6E-04 |                  |
| V-Mom-rotor          | 0.90 | 3.6E-05 | 1.0E-03 |                  |
| W-Mom-rotor          | 0.91 | 4.1E-05 | 9.1E-04 |                  |
| P-Mass-rotor         | 0.94 | 1.2E-05 | 4.7E-04 |                  |
| U-Mom-stator         | 0.89 | 2.1E-05 | 1.1E-03 |       1.3E-02  OK|
| V-Mom-stator         | 0.84 | 9.9E-05 | 3.1E-03 |       3.1E-02  OK|
| W-Mom-stator         | 0.86 | 7.4E-05 | 3.8E-03 |       2.6E-02  OK|
| P-Mass-stator        | 0.89 | 1.4E-05 | 3.8E-04 | 10.0  6.6E-02  OK|
+----------------------+------+---------+---------+------------------+
| H-Energy-rotor       | 0.92 | 7.7E-06 | 2.5E-04 |                  |
| H-Energy-stator      | 0.88 | 5.4E-06 | 1.7E-04 |  6.1  5.4E-02  OK|
+----------------------+------+---------+---------+------------------+
| K-TurbKE-rotor       | 1.62 | 1.2E-04 | 5.7E-03 |                  |
| K-TurbKE-stator      | 0.86 | 1.1E-04 | 3.5E-03 |  6.1  9.7E-02  OK|
+----------------------+------+---------+---------+------------------+
| E-Diss.K-rotor       | 2.52 | 4.0E-04 | 1.6E-02 |                  |
| E-Diss.K-stator      | 0.87 | 1.8E-04 | 4.9E-03 |  7.5  1.5E-03  OK|
+----------------------+------+---------+---------+------------------
```

- Flows across GGI interfaces will be reported in flow summary diagnostics. Changes in GGI flows will occur in various situations. For example, momentum flows change as they undergo rotation at rotational periodic GGI connections. All transport equation flows, including the mass equation, change for the case of pitch change at a frame change GGI connection, as well as the energy equation flows as conservation changes from absolute frame to relative frame energy components. Forces are also reported in the momentum flow balances at contiguous and periodic GGI connections, for cases having non-overlapping portions of the interface.
- Various quantities such as ranges of dependent variables, locations of maximum residuals, etc., are reported at the end of the simulation on a per-domain basis.

```
+-----------------------------------------------------------------------+
|                    Locations of Maximum Residuals                     |
+-----------------------------------------------------------------------+
|       Equation       |  Node # |     X     |     Y     |     Z     |
+-----------------------------------------------------------------------+
```



```
| U-Mom-rotor          |    5109 | 3.849E-01 | 4.761E-02 | 1.572E-01 |
| V-Mom-rotor          |    4501 | 3.846E-01 | 5.065E-02 | 1.176E-01 |
| W-Mom-rotor          |    3085 | 3.802E-01 | 5.029E-02 | 1.177E-01 |
| P-Mass-rotor         |    3642 | 3.799E-01 | 5.260E-02 | 1.572E-01 |
| U-Mom-stator         |   32356 | 4.535E-01 | 5.403E-02 | 5.000E-02 |
| V-Mom-stator         |     496 | 4.541E-01 | 5.343E-02 | 4.939E-02 |
| W-Mom-stator         |     496 | 4.541E-01 | 5.343E-02 | 4.939E-02 |
| P-Mass-stator        |    2275 | 4.536E-01 | 5.349E-02 | 4.949E-02 |
| H-Energy-rotor       |   18527 | 4.535E-01 | 5.694E-02 | 5.263E-02 |
| H-Energy-stator      |     498 | 4.541E-01 | 5.352E-02 | 4.961E-02 |
| K-TurbKE-rotor       |    8583 | 4.110E-01 | 4.671E-02 | 5.692E-02 |
| E-Diss.K-rotor       |    8598 | 4.109E-01 | 4.667E-02 | 5.719E-02 |
| K-TurbKE-stator      |    7264 | 4.540E-01 | 5.358E-02 | 4.959E-02 |
| E-Diss.K-stator      |    7264 | 4.540E-01 | 5.358E-02 | 4.959E-02 |
+--------------------------------------------------------------------+
```

CFX-Solver Output File (Combustion Runs)

Running a combustion simulation in the CFX-Solver is similar to multicomponent fluid runs with the extensions explained in the following.

Multicomponent Specific Enthalpy Diffusion (MCF)

This applies also to non-reacting multicomponent flow: Multicomponent fluids with heat transfer, for which the Unity Lewis Number formulation for MCF energy diffusion is applied, are reported.

```
+--------------------------------------------------------------------+
|          Multi-Component Specific Enthalpy Diffusion               |
+--------------------------------------------------------------------+
Unity Lewis Number (Le=1) for multi-component specific enthalpy
diffusion applied for fluids below.
Domain Name : run
   Mixture
```

Single Step Reactions Heat Release

For single step reactions the heat release per chemical amount of reaction is reported. Numbers are in solver units. Positive numbers indicate exothermic reactions (heat is released) and negative numbers indicate endothermic reactions (energy required for the reaction to occur). The total heat release in the domain resulting from a particular reaction can be computed by multiplying the reactions heat release by its molar reaction rate and integrating over the domain.

```
+--------------------------------------------------------------------+
|                 Single Step Reactions Heat Release                 |
+--------------------------------------------------------------------+
Enthalpy per [mol] of reaction at reference conditions
(Pressure= 1.01325E+05, Temperature= 2.98150E+02)
    HCO Oxygen                                          =  5.5799E+05
    HCN NO Destruction PDF                              =  1.8194E+05
    HCN NO Formation PDF                                =  1.3458E+03
    Reburn NO Fuel Gas PDF                              =  2.7885E+05
    Prompt NO Fuel Gas PDF                              = -9.0298E+04
```

```
  Thermal NO PDF                                          = -1.8060E+05
  Fuel Gas Oxygen                                         =  5.1127E+05
```

Stoichiometric Mixture Fraction

The stoichiometric mixture fraction is reported when a `Flamelet` model or `Burning Velocity` model is used. If the stoichiometric mixture fraction is not specified in the definition of the `Flamelet` library reaction object, the stoichiometric value is reported as `<unknown>`.

```
+--------------------------------------------------------------------+
|                  Stoichiometric Mixture Fraction                   |
+--------------------------------------------------------------------+
Stoichiometric mixture fraction (Zst) for fluids with mixture
fraction combustion models.
Domain Name : run
    Mixture                                          Zst = 5.5000E-02
```

Hydrocarbon Fuel Model: Proximate / Ultimate Analysis

For the hydrocarbon fuel model (the `Hydrocarbon Fuel` option for the **Material** setting) the results of the proximate/ultimate analysis calculation are reported:

- The initial composition of the particles (component mass fractions). These values will be applied when no user-specified initial particle composition is specified (default).
- Molar mass and reference specific enthalpy for the `Volatiles` material that is released to the gas phase. The values reported here by default overrule the data specified in the corresponding material object.
- For single step reactions the relative stoichiometric coefficients, which are derived from the fuel analysis, are reported. These values are applied when the `Fuel Analysis` option is specified in the reaction for the corresponding reactant or product.
- For multiphase reactions the relative mass coefficients, which are derived from the fuel analysis, are reported. These values are applied when the `Fuel Analysis` option is specified in the reaction for the corresponding reactant or product.

```
=======================================================================
                  HC Fuel Proximate/Ultimate Analysis
=======================================================================
Initial mass fractions for particle : HC Fuel
    Ash                                                =  1.2580E-01
    Char                                               =  0.0000E+00
    Raw Combustible                                    =  8.7420E-01
Volatiles material properties : Fuel Gas
    Molar Mass [kg/kmol]                               =      18.234
    Ref. Spec. Enthalpy (p= 1.01325E+05, T= 2.98150E+02) = -7.1827E+06
+---------------------------------------------------------------------+
|              Autocomputed Stoichiometric Coefficients               |
+---------------------------------------------------------------------+
Reaction (volatiles oxidation) : Fuel Gas Oxygen
  Reactants:
    Fuel Gas                                           =  1.0000E+00
    O2                                                 =  1.3558E+00
  Products:
    CO2                                                =  9.3953E-01
    H2O                                                =  1.1267E+00
Reaction (NO reburn) : Reburn NO Fuel Gas PDF
  Reactants:
    Fuel Gas                                           =  3.6879E-01
    NO                                                 =  1.0000E+00
  Products:
```



```
    CO2                                                    =  3.4648E-01
    H2O                                                    =  4.1552E-01
    N2                                                     =  5.0000E-01
  +--------------------------------------------------------------------+
  |                  Autocomputed Mass Coefficients                    |
  +--------------------------------------------------------------------+
  Reaction (devolatilisation) : HC Fuel Devolat HCN
    Reactants:
      HC Fuel.Raw Combustible                              =  1.0000E+00
    Products:
      Gas Mixture HCN NO.Fuel Gas                          =  5.0003E-01
      Gas Mixture HCN NO.HCN                               =  1.1857E-02
      HC Fuel.Char                                         =  4.8812E-01
  Reaction (char oxidation) : HC Fuel Char Gibb HCN
    Reactants:
      Gas Mixture HCN NO.O2                                =  2.6049E+00
      HC Fuel.Char                                         =  1.0000E+00
    Products:
      Gas Mixture HCN NO.CO2                               =  3.5817E+00
      Gas Mixture HCN NO.HCN                               =  2.3164E-02
```

CFX-Solver Output File (Particle Runs)

Particle Transport Equations

This section lists the particle transport equations solved during the solver run for each independent domain.

```
 +--------------------------------------------------------------------+
 |      Particle Transport Equations Solved in This Calculation       |
 +--------------------------------------------------------------------+
Domain Type : Default Domain
   x-Mom-Sand Fully Coupled
   y-Mom-Sand Fully Coupled
   z-Mom-Sand Fully Coupled
   x-Mom-Sand One Way Coupled
   y-Mom-Sand One Way Coupled
   z-Mom-Sand One Way Coupled
```

For additional information, see Solved Equations (p. 35).

Particle Fate Diagnostics

Within the particle transport model, each particle is tracked until one of the abort criteria is satisfied or the particle escapes the domain. Particles may also be aborted if a tracking error is found. Because each particle is tracked from its injection point until some abort criteria is met, it does not influence other particles. Thus, a tracking error for one particle does not necessarily stop the execution of the CFX-Solver run. However, if some fundamental problem is found during tracking, then the flow calculations terminates after printing an error message.

At the end of each particle integration step, a diagnostic summary is generated that indicates the fate of all injected particles as outlined below:

```
 +--------------------------------------------------------------------+
 |                      Particle Fate Diagnostics                     |
 +--------------------------------------------------------------------+
 | Particle type          | Fate type                       Particles |
 +--------------------------------------------------------------------+
 | Sand Fully Coupled     | Entered domain                :       200 |
```

```
|                          | Left domain                    :       200 |
+-------------------------------------------------------------------------+
```

The following fate types may be encountered during particle tracking:

Absorbed by Porous Media

This message indicates the number of particle that have been absorbed by the porous media and is determined by the **Absorption Diameter** setting. For details, see Particle Absorption (p. 138) in the ANSYS CFX-Pre User's Guide.

Continue from Last Time Step (Transient Only)

See Waiting for Next Time Step (Transient only) (p. 56).

Collected on Walls

This message indicates that particles are trapped at the wall because both the parallel and perpendicular restitution coefficients are zero. As a result, the particle tracking is terminated and the remaining momentum of the particles is transferred to the wall as a force. For details, see Restitution Coefficients for Particles (p. 200) in the ANSYS CFX-Solver Modeling Guide.

Entered Domain

This message indicates the number of particles that were injected into the domain either at a boundary or particle injection region.

Exceeded Distance Limit

This message indicates that these particles have exceeded the maximum tracking distance with respect to the particle traveling distance. If a large number of particles show this fate, you may need to increase the maximum tracking distance. For details, see Maximum Tracking Distance (p. 215). Any remaining mass, momentum and energy, which needs to be exchanged with the fluid to equilibrate the aborted particle with the local fluid properties, is exchanged in the current control volume.

Exceeded Integration Limit

This message indicates that these particles have exceeded the maximum number of integration steps with respect to the total number of particle integration steps. If a large number of particles show this fate, you may need to increase the maximum number of integration steps. For details, see Maximum Number of Integration Steps (p. 215). Any remaining mass, momentum and energy, which needs to be exchanged with the fluid to equilibrate the aborted particle with the local fluid properties, is exchanged in the current control volume.

Exceeded Time Limit

This message indicates that these particles have exceeded the maximum tracking time with respect to the particle traveling time. If a large number of particles show this fate, you may need to increase the maximum tracking time. For details, see Maximum Tracking Time (p. 215). Any remaining mass, momentum and energy, which needs to be exchanged with the fluid to equilibrate the aborted particle with the local fluid properties, is exchanged in the current control volume.

Fell Below Minimum Diameter

This message indicates that evaporating or reacting particles are not tracked below a diameter of 1.0E-8 [m] because they are too small to have any effect. The minimum diameter can be controlled by expert parameter `pt minimum diameter`, see Particle-Tracking Parameters (p. 375).

Integration Error

This message indicates that the tracking of these particles was terminated due to an unexpected error in the particle tracking integration. If a small number of particles terminate in this way, it is not generally a cause for concern. Any remaining mass, momentum and energy of the particle is neglected.

Left Domain

This message indicates the number of particles that have escaped from the domain and are no longer tracked. Any remaining particle mass, momentum and energy also escapes the domain with particles. This is the normal abort criterion for particles traveling through inlets, openings and outlets.

Sliding along Walls

This message indicates that particles have stopped as they were hitting the wall below the minimum specified impact angle. For details, see Fluid Values for Walls (p. 118) in the ANSYS CFX-Pre User's Guide.

Waiting for Next Time Step (Transient only)

All particles that are *alive* at the end of a fluid time step get the fate `Waiting for Next Time Step`. When the next fluid time step is performed, then these particles are continued to be tracked and their fate is changed to `Continue from Last Time Step`. So across two time steps, both numbers should be the same.

Transient Particle Diagnostics

The transient particle diagnostics allow you to monitor various quantities, like total mass of particles in the domain, penetration of particles from a given PIR (Particle Injection Region), or user-defined location. For details, see Transient Particle Diagnostics (p. 207) in the ANSYS CFX-Solver Modeling Guide. A typical diagnostics output that is recorded in the CFX-Solver Output file is shown below:

```
+--------------------------------------------------------------------+
|                  Transient Particle Diagnostics                    |
+--------------------------------------------------------------------+

Water Droplets

 Total Particle Mass
    Total Particle Mass                        2.0000E-02
 Penetration from PIR
    Axial Penetration                          1.7655E-01
    Radial Penetration                         1.8561E-01
    Normal Penetration                         6.6197E-02
    Spray Angle                                1.9856E+01
 Penetration from Location
    Axial Penetration                          1.7655E-01
    Radial Penetration                         1.8561E-01
    Normal Penetration                         6.6197E-02
    Spray Angle                                4.1836E+01
```

Particle Convergence History

Coupled particles are solved several times during the simulation and each particle tracking step leads to updated sources for the transport equations of the coupled continuous phase. A convergence criterion for the particle solver is the relative change of the generated sources of two successive particle tracking steps. Therefore, a table containing summary of all coupled particle equations, separately listed for each particle type, is generated at the end of each particle tracking step. This table contains the information on integrated sources as well as the source change rate as outlined below.

```
+--------------------------------------------------------------------+
|   Particle Equation    |  Total source and source change rates     |
+------------------------------------------------------------------+
|                        | Equation            Source      Rate      |
+--------------------------------------------------------------------+
| Domain: PipeValve                                                  |
+--------------------------------------------------------------------+
| Sand Fully Coupled     | x-Mom               4.016E-02   0.0085    |
|                        | y-Mom               7.765E-02   0.0062    |
```

```
|                          | z-Mom                1.261E-01   0.0035    |
+--------------------------------------------------------------------+
```

The `Source` column refers to the new particle source to the ϕ equation, which is calculated as the sum over all vertices of the absolute particle source values to that equation. For details, see Particle Source Change Target (p. 214) and Particle Under Relaxation Factors (p. 214) in the ANSYS CFX-Solver Modeling Guide.

The `Rate` column shows the fractional change in the source between the current and the previous injection. The source change rate can also be graphically monitored in the CFX-Solver Manager.

For additional information, see Convergence Control for Particle Transport (p. 222) in the ANSYS CFX-Solver Modeling Guide.

Integrated Particle Flows

At the end of a run, the integrated particle mass, momentum and energy flows over all boundaries and at all particle injection regions are printed to the CFX-Solver Output file. If the expert parameter MONITOR TOTALS is set to `T`, the same information is printed at the end of each particle tracking step. In a transient run, the time integrated total particle flows are also listed at the end of a run.

CPU Requirements of Numerical Solution

At the end of the run, a summary of the CPU time spent inside of the particle tracking routine is given as outlined in the table below. This information may be used to determine the time spent per particle.

```
+--------------------------------------------------------------------+
|               CPU Requirements of Numerical Solution               |
+--------------------------------------------------------------------+

 Subsystem Name                  Discretization       Linear Solution
                                (secs.   %total)     (secs.   %total)
 ------------------------------------------------------------------------
 Momentum and Mass             1.00E+02  48.2 %     1.79E+01   8.6 %
 TurbKE and Diss.K             3.46E+01  16.6 %     2.17E+01  10.4 %
                               -------- -------     --------  ------
 Subsystem Summary             1.35E+02  64.9 %     3.96E+01  19.1 %

 Particle Tracking             2.18E+01   10.5 %
 Variable Updates              5.91E+00    2.8 %
 File Reading                  6.25E-02    0.0 %
 Miscellaneous                 5.72E+00    2.7 %
                               --------
 Total                         2.08E+02
```

CFX-Solver Output File (ANSYS Multi-field Runs)

An ANSYS Multi-field run can be launched from the CFX-Solver Manager (or from the command line using the `cfx5solve` command) or from the CFX Launcher. Some content is added to the output file for all ANSYS Multi-field runs, and extra content is added in the case of a run launched by CFX.

If the ANSYS input file is processed, then this is noted in the output file, and the location of the new input file which is created is recorded.

```
+--------------------------------------------------------------------+
|                                                                    |
|            Processing ANSYS Input File (Running CCL2MF)            |
|                                                                    |
+--------------------------------------------------------------------+
Created /home3/cfdsjw/tmp/comp/StaticMixer_001.ansys/ANSYS.mf
```

If the ANSYS Solver was started by CFX, then this is also noted in the output file.

```
+--------------------------------------------------------------------+
|                                                                    |
|                        Starting ANSYS Solver                       |
|                                                                    |
+--------------------------------------------------------------------+
```

The host name and port number used by CFX-Solver to communicate to the ANSYS Solver is recorded. If you started only CFX-Solver using the CFX-Solver Manager or the command line, then you will have provided these; otherwise, the port number is determined automatically by the ANSYS Solver and communicated to CFX-Solver by the start-up script.

```
Connecting to the following master process:
     Host Name   : sceptre
     Port Number :  34586
```

Information on what data will be exchanged on what boundaries is recorded so that you can check that all is as you expect it to be.

```
+--------------------------------------------------------------------+
|    Boundary Condition Data Supplied by External Solver Coupling    |
+--------------------------------------------------------------------+
ANSYS Multi-field Solver : ANSYS
     CFX Boundary    : Interface
     CFX Variable    : Total Mesh Displacement
     ANSYS Interface : 1
     ANSYS Variable  : DISP
```

For steady-state runs only, each time a new coupling step begins, it is noted in the output file.

```
====================================================================
|                     COUPLING STEP =    1                         |
====================================================================
```

The output file also notes each time a new stagger iteration begins.

```
--------------------------------------------------------------------
|                  COUPLING/STAGGER ITERATION =    1                |
--------------------------------------------------------------------
```

CFX-Solver Output File (Radiation Runs)

Convergence History

When the `Thermal Radiation P1`, `Discrete Transfer`, or `Monte Carlo` model has been selected, a new variable, `I-Radiation`, is computed.

P1 Model

```
====================================================================
OUTER LOOP ITERATION =    1                    CPU SECONDS = 3.67E+01
--------------------------------------------------------------------
|       Equation       | Rate | RMS Res | Max Res |  Linear Solution |
+----------------------+------+---------+---------+------------------+
...
+----------------------+------+---------+---------+------------------+
```

```
| I-Radiation          | 0.00 | 3.6E-08 | 4.6E-07 |  5.2  3.5E-02  OK|
+----------------------+------+---------+---------+------------------+
```

Discrete Transfer Model

`I-Radiation` data is output every *n*th iteration, where *n* is specified by an `Iteration Interval` parameter (which can be set in **Solver Control** > **Advanced Options** > **Thermal Radiation Control** in CFX-Pre). The row containing this data has its own column headings, which override the table headings just for the row. The column headings are:

- `#Its` is the number of iterations required to obtain a converged radiation solution to within a specified tolerance. This number is usually 1 unless there are reflective boundaries (emissivity less than 1). The default maximum number of iterations and tolerances are 10, and 0.01 respectively. These values can be set in **Solver Control** > **Advanced Options** > **Thermal Radiation Control** > **Ray Tracing**.
- `Vol Chg` is the maximum normalized change in volumetric absorbed radiation at convergence.
- `Sur Chg` is the maximum normalized change in surface absorbed radiation at convergence.
- `%Lost` is the percentage of ray traces lost due to tracking errors, or non-overlap boundaries at domain interfaces. Values greater than 5% are an indication of a setup error.
- `%Imbal` is the percentage imbalance of radiative energy. This value should be 0 or a small value. Otherwise, the results are not reliable.

```
+----------------------+------+---------+---------+------------------+
| I-Radiation          | #Its | Vol Chg | Sur Chg | %Lost  %Imbal    |
|   Gray               |    1 | 0.0E+00 | 0.0E+00 |  0.38    1.94    |
+----------------------+------+---------+---------+------------------+
```

This variable is also included in the `Locations of Maximum Residuals` and `Peak Values of Residuals` tables that appear in the output file.

Monte Carlo Model

`I-Radiation` data is output every *n*th iteration, where *n* is specified by an `Iteration Interval` parameter (which can be set in the **Solver Control** in CFX-Pre on the **Advanced Options** tab). The row containing this data has its own column headings, which override the table headings just for the row. The column headings are:

- `%SD Sur` is the maximum normalized standard deviation of the irradiation flux at an element face on a boundary (`Irradiation Flux.Normalized Std Deviation`).
- `%SD Vol` is the maximum normalized standard deviation of the radiation intensity within a finite element (`Radiation Intensity.Normalized Std Deviation`).
- `%Lost` is the percentage of histories lost due to tracking errors or non-overlap boundaries at domain interfaces. Values greater than 5% are indication of a setup error.
- `%Imbal` is the percentage imbalance of radiative energy. This value should be 0 or a small value. Otherwise, the results are not reliable.

```
+----------------------+------+---------+---------+------------------+
| I-Radiation          |      | %SD Sur | %SD Vol | %Lost  %Imbal    |
|   Full Spectrum      |      | 4.6E+01 | 1.0E+02 |  0.00    0.00    |
+----------------------+------+---------+---------+------------------+
```

The `Variable Range Information` table has the output variable `Radiation Intensity` listed.

Summary Fluxes

- It should be noticed that for heat flux specified boundaries (adiabatic, for example) the specified heat flux can verified by adding q_{rad} to the boundary flow in the H-Energy flow summary.

  $$q_{\text{spec}} = q_{\text{rad}} + q_{\text{cond}} \quad \text{(Eq. 3.3)}$$

- An I-Radiation section is included...


```
+--------------------------------------------------------------------+
|                          I-Radiation                               |
+--------------------------------------------------------------------+
Boundary         : Airin                                  -1.5659E+03
Boundary         : Combustor Default                      -3.5729E+02
Boundary         : Downcomer Wall                         -5.1864E+02
Boundary         : Fuelin                                 -1.0076E+03
Boundary         : Outlet                                  1.6636E+01
Domain           : Combustor                               5.2454E+03
Domain Interface : Domain Interface 1                      5.3004E-01
Domain Interface : Domain Interface 2                     -5.2638E+00
Domain Interface : LowerGGI                               -1.9895E-01
Domain Interface : Periodic                                1.1727E+01
Domain Interface : UpperGGI                                1.6994E-01
                                                          -----------
Domain Imbalance :                                         1.8195E+03
Domain Imbalance, in %:                                     19.2758 %
```

When the Rosseland model is selected:

- No additional equation is solved. Hence, no thermal radiation flow summary will be available.
- The total heat flux is reported by the H-Energy flow summary.
  $q_{\text{rad}} + q_{\text{cond}}$ (Eq. 3.4)
- The CPU Requirements of Numerical Solution table reports the amount of CPU time spent due to the use of a thermal radiation model.

CFX-Solver Results File

The CFX-Solver results file is generated by the CFX-Solver and contains a full description of the flow simulation including:

- Volume mesh
- Flow solution

The CFX-Solver will generate a results file with a name based on the CFX-Solver input file. For example, running the CFX-Solver using the input file named `file.mdef` in a clean working directory will generate a results file named `file_001.res`. Given a clean working directory, the CFX-Solver will generate a results file with a name based on the CFX-Solver input file. This results file contains the calculated solution values at each mesh node in addition to the original information contained in the CFX-Solver input file. The integer index is incremented each time subsequent simulations are executed in the working directory from the same CFX-Solver input file.

> **Note**
>
> CFX-Solver input files must not contain spaces when run with an associated ANSYS input file (inp).

In CFX, there are two additional types of files that may contain results: transient results and backup files. All backup and transient files are placed in a subdirectory beneath the working directory that is named according to the results file. For example, if the working directory contains the results file `file_001.res`, then backup and transient results files are placed in the subdirectory named `file_001`. Backup and transient files all have names with extension .bak and `.trn`, respectively. For details on how to create these results files in CFX-Pre, see Backup Tab (p. 154) in the ANSYS CFX-Pre User's Guide and Transient Results Tab (p. 155) in the ANSYS CFX-Pre User's Guide. For details on how to create backup results files while the run is executing, see Backup Run Command (p. 98). Note that transient results files are only created during transient simulations.

The naming convention for each backup and transient results file is outlined as follows:

- If a file has a name including `_full`, then it is a full results file that contains a complete set of results and the mesh (for example, `32_full.trn` or `32_full.bak`). These files can be loaded into CFD-Post directly if required, and can be to restart a simulation.

- If a file has a name that does not include `_full`, then it is not a full results file and may contain only selected variables and no mesh (for example, `32.trn` or `32.bak`). Do not try to load one of these files directly into CFD-Post; instead, load the ANSYS CFX results file (named `file_001.res` in the example described above) and use the transient selector to switch between the final results and the results contained in these files.
- In general, transient and backup files contain results for the timestep corresponding to the integer prefix in the file name. For example, `32_full.trn` contains results from the 32nd timestep or outer iteration.
- For simulations involving couplers with external solvers (such as ANSYS Multi-field runs), files with a name including `_CS` (such as `3_CS.trn` or `7_CS_full.bak`) may be present. These backup or transient files are indexed by the coupling step rather than the CFX timestep or outer iteration. For example, `3_CS.trn` corresponds to the results at the end of the 3rd coupling step, not necessarily the 3rd timestep (although the two may be coincident). These files are created in the same way as standard transient and backup files in CFX-Pre, but with a frequency of `Every Coupling Step` or `Coupling Step Interval`.

CFD-Post

The CFX-Solver Results file may be used as input to CFD-Post in order to view the results and produce hard copy output. It may also be used to produce files which are suitable for use with other post-processors by using the CFX Export facility. For details, see File Export Utility (p. 131).

CFX-Solver

The CFX-Solver Results file may also be used as input to the CFX-Solver. The solution is used as the initial values field from which to start a further analysis. For details, see Files Used by the CFX-Solver (p. 25).

CFX Radiation File

When using the Monte Carlo or Discrete Transfer radiation models, output information for radiation is written to a results data file named `results.rg[number].dat`. This default file name will be used unless an error occurs when attempting to open the file as a new file for write access, in which case a new name is created by altering the last character (for example, `results.rg2.dab`), and a message with the new name is written to standard output.

A sample `results.rg[number].dat` file is shown below:

```
Zone          Volume     Temperature Refr. B=1   Emiss Co.   Scat Co.    ...
Room$1        1.309E-02   2.951E+02  1.000E+00   1.000E-02   0.000E+00   ...
Room$2        1.104E-02   2.951E+02  1.000E+00   1.000E-02   0.000E+00   ...
.
.
.
Room$1551     2.172E-03   2.951E+02  1.000E+00   1.000E-02   0.000E+00   ...
Room$1552     5.090E-04   2.951E+02  1.000E+00   1.000E-02   0.000E+00   ...

Total heating and cooling    4.130E+02    3.873E+02

Total path length     1.834E+00


Surface data

Zone          Surface                Area      Temperature Rough   Emiss     ...

Room$1        Room Default$1         6.774E-02   2.991E+02 1.00   1.000E+00   ...
Room$1        VentOut$2              3.232E-02   2.951E+02 1.00   1.000E+00   ...
Room$2        VentOut$12             2.532E-02   2.951E+02 1.00   1.000E+00   ...
.
.
.
Room$1551     Room Default$10633     2.166E-02   2.991E+02 1.00   1.000E+00 ...
Room$1552     Room Default$10634     1.376E-02   2.991E+02 1.00   1.000E+00 ...
```



```
Total surface heating and cooling     2.211E+04   2.179E+04

Total non-thermal emission   3.464E+02

Net total absorbed power     3.624E+00    1.609E-06%


Total surface current and absorbed flux:-     9.818E-01   1.834E-02

PROCESS   1
***********

Number of histories      29423

 IWORK =     264568
 CPU time used     6.906
```

CFX Radiation File Contents

A CFX Radiation file contains the volume information, surface information, and some miscellaneous quantities such as the net total heating, parameters which measure the work needed and the CPU time used. Finally, if radiometers have been included, the results for each radiometer will be printed out. Note that the CPU time does not include the time required to compute the radiometer value.

Volume Information

The volume information is given in a table with the following columns:

- `Zone` is the radiation element's internal name
- `Volume` is the volume of the radiation element
- `Temperature` is the temperature of the element
- `Refr. B=1` is the refractive index of the element
- `Emiss Co.` is the emission coefficient (Units: per length)
- `Scat Co.` is the scattering opacity coefficient (Units: per length)
- `Path len.` is the average length when crossing a radiation element
- `Heating` is average volumetric absorbed radiation
- `Emission` is the volumetric emitted radiation
- `(H-E)*VOL` is the amount of radiative energy that shows up as a square term in the energy equation
- `Error %` is the standard deviation of the path length.

For non-grey models the emission and scattering coefficients are the spectrum integrated coefficients. In a grey model the absorption coefficient equals the emission coefficient, in a non-grey model it can be obtained from the intensity and the heating.

The next five columns contain the results. The seventh is the total path length of photons in the zone (Monte Carlo) or the mean radiation intensity (discrete transfer). The eighth is the heating per unit volume, the ninth the cooling per unit volume and the tenth the net total radiative heating rate for the zone. The last column gives the statistical percentage error for Monte Carlo or the number of samples (rays traced through the zone) used for the heating quadrature in the discrete transfer case.

Finally comes the total heating and cooling rates for the entire volume and, for Monte Carlo, the total mean path length of photons in the geometry. Non-grey models will also have the band by band cooling.

Surface Information

In this output the first column is the zone name and the second the surface name. The third is the surface temperature, the fourth the surface roughness and the fifth is the surface emissivity, spectrum integrated in the case of a non-grey

model. Grey surfaces have an albedo equal to one minus the emissivity, in the non-grey case the integrated albedo can be obtained from the surface heating and the incident flux (see).

The next five columns contain the results. The sixth is the surface current (Monte Carlo) or average incident radiation flux (discrete transfer). The seventh is the heating per unit area, the eighth the cooling per unit area and the ninth the net total radiative heating. The last column is the statistical error (Monte Carlo) or the number of surface nodes used for sampling (discrete transfer).

Finally comes the total heating and cooling rates for all the surfaces in the geometry. Non-grey models will also show the band by band cooling.

After the above information, the overall net heating for the model is printed. This is a measure of how good the calculation was since this figure should be zero. For a Monte Carlo calculation the total surface current and absorbed photon flux is printed, these figures should sum to unity and can therefore be used as another measure of the accuracy of the calculation. Next the number of histories computed (Monte Carlo) or the number of angular ordinates (Discrete transfer) is printed. The last numbers are the work estimator and the CPU time. The work estimator is defined in terms of units of work where a unit of work is the computational effort to trace a photon (ray) to the next event (surface) and process that event (update the recursion relation) for Monte Carlo (Discrete transfer). If radiometers have been calculated, the angular calibration table that is used will be printed; then for each radiometer location, the following will be written: location, direction, temperature, flux.

CFX Partition File

The CFX partition file is generated by the CFX-Solver and used as input for a parallel run. The partition file is used to store information about the partitioning of the mesh. The partition file can be used to view mesh partitions before running the CFX-Solver. To do this, set the expert parameter `write partition number=t` before partitioning, then combine the partition file with the CFX-Solver input file that was used to produce the partition file.

On UNIX systems, type:

```
cat filename.def filename_001.par > newfilename.res
```

On Windows systems, type:

```
copy /b filename.def + filename_001.par newfilename.res
```

This creates a results file that can be loaded in CFD-Post and contains the variable `Real partition number`.

> **Note**
>
> A partition file generated in ANSYS CFX 11.0 or earlier versions is not supported in ANSYS CFX 12.0. If such a file is used in ANSYS CFX 12.0, then an error message is generated.

Additional Files for ANSYS Multi-field Runs

For an ANSYS Multi-field run, the CFX-Solver does not produce any extra files. However, if the ANSYS Solver is launched by CFX, a new directory with a name of the form `<Run Name>.ansys` is created. This is used as the ANSYS working directory and many files will be created here by ANSYS. In addition, the CFX processing of the ANSYS files produces two extra files:

- Components File - this has a name of the form `*.cm` and can be found in the ANSYS working directory. It is created to contain some region definitions for CFD-Post when it reads the ANSYS results files. For details, see ANSYS Files (p. 74) in the ANSYS CFD-Post User's Guide.
- ANSYS Multi-field input file - this has a name of the form `*.mf` and can be found in the ANSYS working directory. It is created whenever you ask for the provided ANSYS input file to be processed as part of defining a run. It contains the ANSYS setup (solid physics) together with the multi-field commands that are constructed from the ANSYS Multi-field settings in the CFX-Solver input file. For details, see Processing the ANSYS Input File (p. 21).

CFX Multi-Configuration Output File

Multi-configuration simulations involve the sequential and sometimes repeated execution of defined configurations. The CFX-Solver generates a multi-configuration output file to summarize the global problem definition and execution of configurations.

The CFX-Solver will generate a multi-configuration output file with a name based on the CFX-Solver Input file. For example, running the CFX-Solver using the input file named `file.mdef` in a clean working directory will generate the output file named `file_001.out`.

Header

The header is written at the start of every multi-configuration output file and contains information regarding the command which started the simulation. This information is useful to check which files were used to start the run.

CFX Command Language for the Run

The CFX Command Language section describes the global (that is, simulation level) problem, which includes library definitions (such as materials, expressions, etc…) and simulation control specifications (such as, configuration definitions).

The section for the command file looks similar to the following:

```
+--------------------------------------------------------------------+
 |                                                                    |
 |                     CFX Command Language for Run                   |
 |                                                                    |
 +--------------------------------------------------------------------+



LIBRARY:
  MATERIAL: Air at 25 C
    Material Description = Air at 25 C and 1 atm (dry)
    Material Group = Air Data, Constant Property Gases
    Option = Pure Substance
    Thermodynamic State = Gas
    PROPERTIES:
      Option = General Material
      EQUATION OF STATE:
        Density = 1.185 [kg m^-3]
        Molar Mass = 28.96 [kg kmol^-1]
        Option = Value
      END
      SPECIFIC HEAT CAPACITY:
        Option = Value
        Specific Heat Capacity = 1.0044E+03 [J kg^-1 K^-1]
        Specific Heat Type = Constant Pressure
      END
      REFERENCE STATE:
        Option = Specified Point
        Reference Pressure = 1 [atm]
        Reference Specific Enthalpy = 0. [J/kg]
        Reference Specific Entropy = 0. [J/kg/K]
        Reference Temperature = 25 [C]
      END
      DYNAMIC VISCOSITY:
        Dynamic Viscosity = 1.831E-05 [kg m^-1 s^-1]
        Option = Value
      END
```

```
      THERMAL CONDUCTIVITY:
        Option = Value
        Thermal Conductivity = 2.61E-02 [W m^-1 K^-1]
      END
      ABSORPTION COEFFICIENT:
        Absorption Coefficient = 0.01 [m^-1]
        Option = Value
      END
      SCATTERING COEFFICIENT:
        Option = Value
        Scattering Coefficient = 0.0 [m^-1]
      END
      REFRACTIVE INDEX:
        Option = Value
        Refractive Index = 1.0 [m m^-1]
      END
      THERMAL EXPANSIVITY:
        Option = Value
        Thermal Expansivity = 0.003356 [K^-1]
      END
    END
  END
END
SIMULATION CONTROL:
  CONFIGURATION CONTROL:
    CONFIGURATION: Both Boxes Transient
      Flow Name = Copy of Both Domains
      ACTIVATION CONTROL:
        CONTROL CONDITION: After Top Box Steady State
          Configuration Name List = Top Box Steady Only
          Option = End of Configuration
        END
      END
      CONFIGURATION EXECUTION CONTROL:
        INITIAL VALUES SPECIFICATION:
          INITIAL VALUES CONTROL:
            Use Mesh From = Solver Input File
          END
          INITIAL VALUES: Initial Values 1
            Configuration Name = Top Box Steady Only
            Option = Configuration Results
          END
        END
        PARTITIONER STEP CONTROL:
          Multidomain Option = Independent Partitioning
          Runtime Priority = Standard
          EXECUTABLE SELECTION:
            Double Precision = No
          END
          PARTITIONING TYPE:
            MeTiS Type = k-way
            Option = MeTiS
            Partition Size Rule = Automatic
          END
        END
        SOLVER STEP CONTROL:
          Runtime Priority = Standard
          EXECUTABLE SELECTION:
```

```
        Double Precision = On
      END
      PARALLEL ENVIRONMENT:
        Start Method = HP MPI Local Parallel
      END
    END
  END
  CONFIGURATION: Top Box Steady Only
    Flow Name = Top Domain Steady State
    ACTIVATION CONTROL:
      CONTROL CONDITION: Cyclical Activation
        Configuration Name List = Both Boxes Transient
        Option = End of Configuration
      END
      CONTROL CONDITION: Start of Sim
        Option = Start of Simulation
      END
    END
    CONFIGURATION EXECUTION CONTROL:
      INITIAL VALUES SPECIFICATION:
        INITIAL VALUES CONTROL:
          Continue History From = Initial Values 1
          Use Mesh From = Solver Input File
        END
        INITIAL VALUES: Initial Values 1
          Configuration Name = Both Boxes Transient
          Option = Configuration Results
        END
      END
      PARTITIONER STEP CONTROL:
        Multidomain Option = Independent Partitioning
        Runtime Priority = Standard
        EXECUTABLE SELECTION:
          Double Precision = No
        END
        PARTITIONING TYPE:
          MeTiS Type = k-way
          Option = MeTiS
          Partition Size Rule = Automatic
        END
      END
      SOLVER STEP CONTROL:
        Runtime Priority = Standard
        EXECUTABLE SELECTION:
          Double Precision = On
        END
        MEMORY CONTROL:
          Memory Allocation Factor = 2
        END
        PARALLEL ENVIRONMENT:
          Start Method = HP MPI Local Parallel
        END
      END
    END
  END
END
TERMINATION CONTROL:
```

```
    CONTROL CONDITION: Control Condition 1
      Configuration Name = Top Box Steady Only
      Maximum Number of Configuration Steps = 2
      Option = Maximum Number of Configuration Steps
    END
  END
  EXECUTION CONTROL:
    RUN DEFINITION:
      Solver Input File = mcfg1.mdef
    END
  END
END
```

Simulation Execution Summary

The body of the multi-configuration output file summarizes the configuration execution sequence. The output is divided into simulation and configuration steps.

At each simulation step, the simulation step counter is incremented and the activation conditions for all configurations are evaluated. All active configurations are executed, but in no particular order. Once all active configurations have finished their execution for the current simulation step, a new step begins. Subsequent simulation steps are executed until any of the following occurs:

- there are no active configurations in a simulation step
- any simulation termination control conditions are satisfied
- the user stops the simulation while in progress by using **Stop Current Run** in the multi-configuration simulation workspace, or by executing the **cfx5stop** command for the simulation run directory.

Additional details regarding configuration activation conditions and simulation termination controls are provided in Configurations (p. 221) in ANSYS CFX-Pre User's Guide and Termination Control (p. 219) in ANSYS CFX-Pre User's Guide, respectively.

A new configuration setup occurs, and the configuration step counter is incremented, each time a given configuration is executed. Configuration step output includes the following information:

- the configuration step counter value
- the name of the configuration
- a listing of all configuration activation conditions that were satisfied
- the status at the end of the run (e.g. whether it succeeded or failed, or if the user stopped the run in progress)
- notes regarding any CFX Command Language (CCL) changes made using **Edit Run in Progress** during the run.

A sample simulation execution summary is provided below. This summary presents the execution of three simulation steps.

1. Only the **Top Box Steady Only** configuration was active and ran. This configuration was activated by the **Start of Sim** condition defined in **Simulation Control** > **Configuration** > **General Settings** > **Activation Condition(s)** and ran successfully.
2. Only the **Both Boxes Transient** configuration was active and ran. This configuration was activated by the **After Top Box Steady State** condition, and ran successfully. During this run, changes to both `LIBRARY` and `FLOW CCL` were made using **Edit Run in Progress**, and the included notes indicate how these changes will be propagated to subsequent configuration runs.
3. On the **Top Box Steady Only** configuration was active and ran. Notice that the configuration step counter for this run indicates that this is the second time this configuration has been run, and that the configuration was activated by the **Cyclical Activation** condition. This run also completed successfully.

```
+--------------------------------------------------------------------+
|                                                                    |
|                        Simulation Execution                        |
|                                                                    |
```



```
+----------------------------------------------------------------+


================================================================
SIMULATION STEP =        1
----------------------------------------------------------------
|  Config.   | Configuration Name                                |
|   Step     | * Execution Details                               |
+-----------+-------------------------------------------------+
|        1   | Top Box Steady Only                               |
|            | * Act. by : Start of Sim                          |
|            | * Status  : Succeeded                             |
+-----------+-------------------------------------------------+

================================================================
SIMULATION STEP =        2
----------------------------------------------------------------
|  Config.   | Configuration Name                                |
|   Step     | * Execution Details                               |
+-----------+-------------------------------------------------+
|        1   | Both Boxes Transient                              |
|            | * Act. by : After Top Box Steady State            |
|            | * Status  : Succeeded                             |
|            | * NOTE    : The LIBRARY definition changed during this |
|            |             run.  These changes will be applied to all |
|            |             remaining configuration runs.         |
|            | * NOTE    : The FLOW definition changed during this  |
|            |             run.  These changes will be applied to all |
|            |             remaining runs of this configuration.  |
+-----------+-------------------------------------------------+

================================================================
SIMULATION STEP =        3
----------------------------------------------------------------
|  Config.   | Configuration Name                                |
|   Step     | * Execution Details                               |
+-----------+-------------------------------------------------+
|        2   | Top Box Steady Only                               |
|            | * Act. by : Cyclical Activation                   |
|            | * Status  : Succeeded                             |
+-----------+-------------------------------------------------+
```

Simulation Termination Condition Summary

This section of the multi-configuration output file identifies the reason(s) for terminating the simulation. In the example provided below, the user defined condition named **Control Condition 1** was satisfied. For information regarding how this condition was defined, refer to the discussion in Termination Control (p. 219) in ANSYS CFX-Pre User's Guide.

```
================================================================
 TERMINATION CONDITION SUMMARY
----------------------------------------------------------------
```

```
* User: Control Condition 1
```

CFX Multi-Configuration Results File

The CFX-Solver will generate a results file with a name based on the CFX-Solver Input file. For example, running the CFX-Solver using the input file named `file.mdef` in a clean working directory will generate the multi-configuration results file named `file_001.mres`.

For multi-configuration simulations, the CFX-Solver Input and CFX-Solver Results files are very similar; both are small files that contain only high level information about the simulation. In particular, the results file contains:

- global information about the simulation's overall definition (such as `Library CCL`, configuration definitions and sequencing, etc…)
- information about the configuration files (`*.cfg` which contain configuration specific settings and meshes)
- information about the CFX-Solver Results files (for details, see CFX-Solver Results File (p. 60)) generated for each configuration step.

CFX-Solver Output and CFX-Solver Results files for each configuration (and configuration step) executed are contained within the multi-configuration simulation's run directory. For example, running the CFX-Solver using the input file named `file.mdef` in a clean working directory will generate a file and directory structure similar to the following:

- file_001.mres
- file_001.out
- file_001/
  - Configuration1_001
  - Configuration1_001.out
  - Configuration1_001.res
  - Configuration1_002.out
  - Configuration1_002.res
  - Configuration2_001
  - Configuration2_001.out
  - Configuration2_001.res

In this case, the directories `file_001/Configuration1_001` and `file_001/Configuration2_001` each contain the transient and backup files created during the respective configuration runs.

Chapter 4. Residual Plotting

CFX-Solver calculates solutions to various equations. However, many cases result in residual values. This is due to an equation not being fully satisfied. Of course, if the solution is exact, then the residual is zero. However, because equation results only approximate physics, the results in a solution do not always match reality.

This chapter describes:

Equation Residual

CFX-Solver calculates the solution to various equations, given the appropriate boundary conditions and models for your particular CFD problem. For details, see Governing Equations (p. 17) in the ANSYS CFX-Solver Theory Guide.

At any stage of a calculation, each equation will not be satisfied exactly, and the “residual” of an equation identifies by how much the left-hand-side of the equation differs from the right-hand-side at any point in space. If the solution is “exact,” then the residual is zero. Exact means that each of the relevant finite volume equations is satisfied precisely. However, because these equations only model the physics approximately, this does not mean that the solution exactly matches what happens in reality. If a solution is converging, residuals should decrease with successive timesteps.

For example, assume that a given residual is 0.0005 kg s^-1. It is not obvious whether such a residual is large or small. For instance, if the problem involves flows such that about 0.5 kg flows into (and out of) each mesh element every second, then a residual of 0.0005 kg s^-1 means the equation is satisfied to within one part in a thousand, which is likely a reasonable solution. However, if the problem involves flows of about 0.001 kg s^-1 into each mesh element, then the residual is nearly as large as the flow, and the solution is not good.

To make the scales of the residuals meaningful, the solver normalizes values by dividing the appropriate scales at each point. Solver Manager plots these normalized residuals using a log (base 10) scale.

The exact details of how the residuals are normalized are involved. For details, see Residual Normalization Procedure (p. 254) in the ANSYS CFX-Solver Theory Guide. However, it is useful to know that residuals are divided by the solution range. If linear solution diverges, this range may be very large and the normalized residuals would be meaningless.

Convergence Results and RMS

A measure of how well the solution is converged can be obtained by plotting the residuals for each equation at the end of each timestep. A reasonably converged solution requires a maximum residual level no higher than 5.0E-4. Typically, the RMS (Root Mean Square) residual will be an order of magnitude lower than this.

The RMS residual is obtained by taking all of the residuals throughout the domain, squaring them, taking the mean, and then taking the square root of the mean. This should present an idea of a typical magnitude of the residuals.

The Maximum Residuals and/or the RMS Residuals can be displayed in the convergence history plots by selecting a specific monitor in **Monitor Settings**. For details, see Monitors Tab (p. 92).

The increase of a residual after any particular timestep does not imply that the solution is diverging. It is usual for residuals to occasionally get larger, especially at the beginning of a run.

Note that even though convergence is good, there are still places where the residuals become larger temporarily.

It is also possible to have runs that do not converge at all, but simply deviate around the same values.

If the solution fails to converge, or convergence is happening only very slowly, some tips on how to improve the convergence are available. For details, see Monitoring and Obtaining Convergence (p. 336) in the ANSYS CFX-Solver Modeling Guide.

Tip

If you want to obtain residual plots for old runs, select **File** > **Monitor Finished Run** and select a file to view.

Transient Residual Plotting

When monitoring a transient run with the plotting of coefficient loop data selected, CFX-Solver Manager outputs the monitor data for each coefficient loop (cloop) within each timestep. Each timestep is divided by the number of inner coefficient loops. The values are produced for all variables in each cloop.

With the plotting selected, CFX-Solver Manager outputs the graph of monitor data, plotting a point for each coefficient loop evenly across the space between the previous timestep and the current one, so that the final coefficient loop datum is plotted on the X coordinate of the current timestep. For details, see Global Plot Settings Tab (p. 97).

Example: Assume that four coefficient loops were run for timestep 39. The first cloop value for a plotted variable appears at position 38.25, the second at 38.5, the third at 38.75, and the fourth and final value is plotted at 39.

Residual Plotting for ANSYS Multi-field Runs

The CFX-Solver Manager can be used to launch an ANSYS Multi-field Run. For such a run, it is not sufficient to consider only the convergence of CFX-Solver when deciding if results are converged: in addition, the convergence of CFX-Solver and the convergence of the data exchanged between the CFX and ANSYS Solvers must also be considered. The CFX-Solver Manager shows extra plots when monitoring an ANSYS Multi-field run to allow a graphical display of these extra convergence quantities.

For details, refer to the following sections:

- Coupling CFX to an External Solver: ANSYS Multi-field Simulations (p. 295) in the ANSYS CFX-Solver Modeling Guide
- ANSYS Field Solver Plots (p. 72)
- ANSYS Interface Loads Plots (p. 73)

For the extra plots to be displayed, the CFX-Solver Manager must have started CFX-Solver by selecting an **MFX Run Mode** of either `Start ANSYS and CFX` or `Start ANSYS only` (see MultiField Tab (p. 9)), or CFX-Solver must have been started from the command line with the `cfx5solve` command. It is not possible for CFX-Solver Manager to display these plots if CFX-Solver was started by the CFX Launcher or from the command line directly.

ANSYS Field Solver Plots

These plots are only produced when the solid physics is set to use large displacements or when other non-linear analyses are performed. It shows convergence information from CFX-Solver. Full details of the quantities plotted are described in the ANSYS user documentation, in "*Newton-Raphson Procedure*".

In general, the CRIT quantities are the convergence criteria for each relevant variable, and the L2/L1/INF quantities represent the L2 Norm/L1 Norm/Infinite Norm (respectively) of the relevant variable (which one is available for each variable depends on the norm chosen using the `CNVTOL` command, and will default to L2 if the ANSYS input file was produced by Simulation). The convergence criteria can change during a run as the default criteria is calculated using a reference value for each quantity which is updated as the solution progresses. For convergence, the L2, L1, or INF value should be below the criteria.

The x-axis of the plot is the cumulative iteration number for ANSYS, which does not correspond to timesteps, coupling steps, or stagger iterations. Several ANSYS iterations will be performed for each timestep or coupling step, depending on how quickly ANSYS converges. You will usually see a somewhat spiky plot, as each quantity will be unconverged at the start of each time or coupling step, and then convergence will improve.

Several ANSYS Field Solver plots may be present, depending on the physics being solved within ANSYS.

ANSYS Interface Loads Plots

These plots show the convergence for each quantity which is part of the data exchanged between the CFX and ANSYS Solvers. There will always be two plots: ANSYS Interface Loads (Structural) and ANSYS Interface Loads (Thermal). The structural plot contains convergence information on forces and displacements, and the thermal plot contains information on temperature and heat flows/fluxes.

For each variable (each x-, y- and z-component of the load is a separate variable), the convergence norm for the data transferred across the interface is given by:

$$\Phi = \frac{\sqrt{\sum (u_{new} - u_{old})^2}}{\sqrt{\sum u_{new}^2}} \qquad \text{(Eq. 4.1)}$$

where Φ represents the L2 norm of the transferred load, u_{old} is the load component transferred at the last stagger iteration, u_{new} is the load component transferred at this stagger iteration, and the sum is over all the individual load component values transferred (at different points in space). Each quantity is considered to be converged when $\Phi < \Phi_{\text{min}}$, where Φ_{min} is the convergence target for that quantity set in CFX-Pre (see Solver Controls, External Coupling Tab (p. 300) in the ANSYS CFX-Solver Modeling Guide) or directly by the multi-field commands in the ANSYS input file (`MFCO` command).

> **Note**
>
> The definition of the convergence norm in Equation 4.1 (p. 73) was changed in Release 12.0. To achieve convergence in Release 12.0 (and later) that is similar to that achieved prior to Release 12.0, set the new convergence target to be the square root of the convergence target used prior to Release 12.0. Furthermore, the default value for the convergence target in Release 12.0 was changed to 0.01.

Convergence of each quantity transferred across the interface is reported as e, where:

$$e = \frac{\log(\Phi/\Phi_{\text{min}})}{\log(10/\Phi_{\text{min}})} \qquad \text{(Eq. 4.2)}$$

and this is the quantity plotted on the ANSYS Interface plots. This implies that each quantity has converged when the reported convergence reaches a negative value. In general, the ANSYS Interface Loads (Structural) plot will contain six lines, corresponding to three force components (FX, FY, and FZ) and three displacements (UX, UY, and UZ), and the ANSYS Interface Loads (Thermal) plot will contain two lines (temperature and heat flow/flux). However, if the simulation is effectively 2D (only one element across), then forces and displacements in the third dimension are not exchanged and the corresponding lines will not be present.

The x-axis of the plot corresponds to the cumulative number of stagger iterations (coupling iterations) and there are several of these for every timestep. A “spiky” plot is expected as the quantities will not be converged at the start of a timestep.

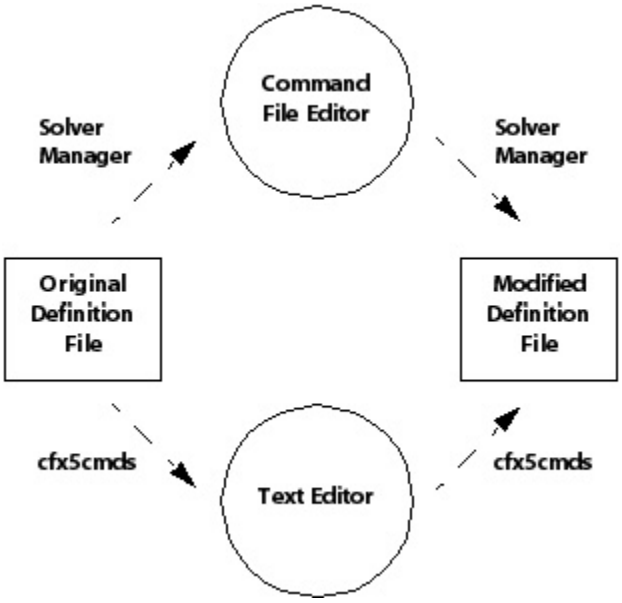
Chapter 5. Editing CFX-Solver Input Files

This chapter discusses the details of editing existing CFX-Solver input (or results) files to make minor changes to a CFD model, without having to use CFX-Pre:

Workflow Overview

Modifications to CFX-Solver input (results) files can be performed using either the **Command Editor** dialog box in CFX-Solver Manager or the `cfx5cmds` command from the command line, as shown in Figure 5.1, "CFX-Solver Input File Modification Workflows" (p. 75).

Figure 5.1. CFX-Solver Input File Modification Workflows



Command File Editor Overview

The simplest method of making changes to an existing CFX-Solver input file is to use the Command File editor. The Command File editor provides a tree-structured representation of a CFX-Solver input file. This allows modification of parameter settings and the addition of new parameters to existing CFX-Solver input files as required. Edit CFX-Solver File Command (p. 101)

The Command File editor can be invoked in multiple ways and each way provides a different function:

- **Tools** > **Edit CFX-Solver File**

  Edit CFX-Solver File Command (p. 101)

- **Tools** > **Edit Run In Progress**

  For details, see Edit Run In Progress Command (p. 105).

- **Tools** > **Edit Current Results File**

  For details, see Edit Current Results File Command (p. 105).

Menus in the Command File Editor

The Command File editor has three menus:

File Menu

The **File** menu contains:

Save Command

Saves the file and returns the CFX Command Language file information to the CFX-Solver input file.

Validate Command

Checks the format of the CFX Command Language file for necessary content and reports any errors. Validation of units is performed each time a parameter is edited.

Exit Command

Closes the Command File editor. A prompt is displayed if there are unsaved changes.

Edit Menu

The **Edit** menu contains:

Add Parameter Command

Some categories (e.g., the `FLOW` > `SOLVER CONTROL` > `CONVERGENCE CONTROL` section) allow additional parameters that are not shown by default.

The **Add Parameter** option is used to:

- Add a new parameter to a category when the category is selected.
- Add an expert parameter to the CFX Command Language file after the Expert Parameter Section has been created. For details, see Add Expert Parameter Section Command (p. 78).

If a parameter can be added to a section, a dialog box displays with a drop-down list of available parameters. Ensure that parameters being added make sense and use correct units as required.

1. Select a parameter that allows additional parameters to be added.
2. Select **Edit** > **Add Parameter**.

3. In the drop down list select a parameter.
4. Under Value, enter specific information about the parameter.

   For details, see Command File Editor Rules (p. 78).

5. Click **OK**.

Note that it is also possible to add a parameter that is not in the drop-down list by typing its name into the top field and then entering a value. The CFX-Solver will subsequently fail if either the supplied name or value is inappropriate.

Edit Parameter Command

The **Edit Parameter** option allows changes to an existing expert parameter in the CFX Command Language file.

When the Command File editor is first opened, the Root is displayed. It contains three categories: `LIBRARY`, `EXECUTION CONTROL` (seen only when editing version 5.5 or later `.res` files) and `FLOW`.

In .def files, only two categories are displayed: `LIBRARY` and `FLOW`.

> **Note**
>
> Parameters that can be edited display the value in green. Other parameters cannot be changed from the Command File editor.

Expanding categories

Additional information about a category can be seen by expanding it.

- Click the plus/minus boxes to expand or reduce the category selection.

> **Tip**
>
> Right-click to expand the category and all of its subcategories.

Editing entries

> **Important**
>
> Editing the CFX Command Language file changes the CFX-Solver input file, but does not make changes to the CFX-Pre case file. In other words, changes made by the Command File editor are used by the CFX-Solver but do not appear when the case file is reopened in CFX-Pre.

1. Expand the **Name** of the entry until a **Value** in green or orange is displayed.

   Editable values display in green or in orange.

2. Double-click the value to edit.

   The **Edit Parameter** dialog box is displayed.

3. Edit the value as required.

   In some parameters this may require edits to text and in others a selection in a drop-down list box. For details, see Command File Editor Rules (p. 78).

4. Click **OK**.
5. When all modifications have been made, save the file.

   Changes made to the CFX Command Language file are written in the CFX-Solver input file.

Delete Parameter Command

The **Delete Parameter** option is used to remove an expert parameter from the CFX Command Language file.

Add Expert Parameter Section Command

The **Add Expert Parameter Section** provides access to the expert parameters in CFX. An expert parameter section must be created before adding expert parameters to the CFX Command Language file.

Before adding any expert parameters to the CFX Command Language file, first add an expert parameters section to the tree structure displayed in the Command File editor.

1. Select **Edit** > **Add Expert Parameter Section**.

   The `EXPERT PARAMETERS:` section is added to the bottom of the `FLOW` branch of the tree.

2. Expand `FLOW`.
3. Select `EXPERT PARAMETERS`.
4. Add parameters as required.

   For details, see Add Parameter Command (p. 76).

Once the value for an expert parameter has been set, it can be edited as needed. For details, see Edit Parameter Command (p. 77).

Find Command

Allows searching of the CFX Command Language file for a keyword or keywords.

Find Next Command

Finds the next occurrence of the keyword or keywords.

Help Menu

The **Help** menu contains:

- Help On CFX-5 Definition File Editor Command (p. 78)
- Help On Expert Parameters Command (p. 78)

On CFX-5 Definition File Editor Command

The help associated with the Command File editor is launched.

On Expert Parameters Command

The CFX-Solver Expert Control Parameters help is launched.

Command File Editor Rules

If unsure as to an appropriate value to enter, select **Help** > **On Expert Parameters** from the main Command File editor window to view a description and valid values for the parameters.

The following rules apply when using the Command File editor:

- Everything is case-sensitive. Use care to distinguish upper case letters from lower case letters.
- Parameter names must start with a letter (not a number or symbol). Subsequent characters can be letters, numbers, spaces or tabs.
- Spaces appearing before or after a name are not considered to be part of the name.
- Multiple spaces and tabs appearing inside a name are treated as a single space.
- For parameters requiring a logical value, enter `T` or `t` for true, and `F` or `f` for false. For many parameters requiring an integer value only a few integer values are valid.

These rules should suffice for simple editing operations. To perform more complicated editing operations, consider editing the command file directly. For details, see Editing the Command Language (CCL) File (p. 79).

Command File Editor Appearance

Some of the file information under the **Name** and **Value** headings may be truncated when the tree structure is expanded. The amount of space allocated to the name or the value can be configured.

1. Position the mouse cursor between the **Name** and **Value** headings.

   That is, place the cursor over the line that separates the two headings. The cursor appears as a double line with an arrow to the left and the right.

2. Click and drag to expand or contract the width of the column.

   For lengthy lines (such as those that contain lists), double-click on the line. This opens the line in a configurable dialog box for viewing and editing.

Editing the Command Language (CCL) File

In some circumstances, more significant changes may be required to the CFX-Solver input file than the Command File editor allows. CFX allows the use of a text editor to edit the CFX Command Language file.

> **Note**
>
> This feature is for expert users only. Extreme care must be taken when editing a CFX Command Language file. Changes made to the CFX Command Language file are reflected in the CFX-Solver input file and may have negative effects on the model. It is strongly advised that original files are backed up before edits are made to them.

To generate the CFX Command Language file, use the `cfx5cmds` command in a UNIX terminal or a Windows command line. Ensure this is set up correctly to run CFX commands. For details, see Command Line (p. 3) in the ANSYS CFX Reference Guide.

Command Language File Rules

The following rules apply when using a text editor to change a CFX Command Language file:

- Changing the solution units outside of CFX-Pre is not recommended. For details, see Solution Units (p. 143) in the ANSYS CFX-Pre User's Guide.
- Everything apart from expert parameter names is case-sensitive. Use care to distinguish upper case letters from lowercase letters. It is recommended that lower case letters are used for expert parameter names to match those shown in the Command File Editors expert parameters section.
- The name of any variable must start with a letter (not a number or symbol). Subsequent characters can be letters, numbers, spaces, or tabs.
- Spaces appearing before or after a name are not considered to be part of the name.
- Multiple spaces and tabs appearing inside a name are treated as a single space.
- Nothing is sensitive to indentation. Indentation is used only to make the appearance of the text clearer.
- The comment character is the pound, or number sign (#). Anything appearing to the right of this character is ignored. For instance:

  ```
  PARAMETER = 3.2   # This text is ignored
  ```

- The line continuation character is the backslash (\). To break a long line of text into two or more lines, insert a backslash character at the end of the first line. For instance, both of the following lines are handled the same way:

  ```
  NAME = default temperature used in the first simulation
  NAME = default temperature used in the first simulation
  ```

- A line containing CFX Expression Language must be no more than 256 characters long. Other lines can be of any length.

The RULES and VARIABLES files can be a useful guide when editing a CCL file. For details, see RULES and VARIABLES Files (p. 80).

Some objects in the CCL file can be defined with a default value and then overridden by local values. For example, the SOLVER CONTROL section of a CCL file may look like:

```
SOLVER CONTROL :
 CONVERGENCE CONTROL :
  Maximum Number of Iterations = 100
  Timescale Control = Physical Timescale
  Physical Timescale = 5.E-1 [s]
 END
 EQUATION CLASS : momentum
  CONVERGENCE CONTROL :
   Timescale Control = Physical Timescale
   Physical Timescale = 1.E-1 [s]
  END
 END
 CONVERGENCE CRITERIA :
  Residual Type = RMS
  Residual Target = 1.E-4
 END
 ADVECTION SCHEME :
  Option = Upwind
  END
 DYNAMIC MODEL CONTROL :
  Global Dynamic Model Control = No
 END
END
```

The first CONVERGENCE CONTROL object defines values that apply to all equation classes. The physical time scale is then locally overridden for the momentum equation class. Other objects, including the ADVECTION SCHEME, can be locally overridden in the same way. The order in which objects appear is not important, so an object can be assigned an override before its default has been set. Only some parameters may be set to a different local value; for example, it does not make sense to set the Maximum Number of Iterations locally.

RULES and VARIABLES Files

The RULES and VARIABLES files can be useful when editing a CCL file. They provide information on valid options, variables, and dependencies. Both files are located in <CFXROOT>/etc/ and can be viewed in any text editor.

> **Note**
>
> In some cases, the RULES and VARIABLES files contain information about variables, parameters, models and options that are not yet fully supported in the CFX-Solver. Use caution with features that are not documented elsewhere as these may cause the CFX-Solver to fail or produce invalid results.

VARIABLES File

The VARIABLES file lists all the variables available in the CFX-Solver. Information provided for each variable appears similar to the following:

```
VARIABLE: vel
 Option = Definition
 MMS Name  = VEL
 Long Name = Velocity
```

```
 Tensor Type = VECTOR
 Quantity = Velocity
 Status = P
 User Level = 1
 Output to Jobfile = No
 Output to Postprocessor = Yes
 Component Short Names = \
  u, \
  v, \
  w
 Component Long Names = \
  Velocity u, \
  Velocity v, \
  Velocity w
 Component MMS Names = \
  VEL-1, \
  VEL-2, \
  VEL-3
 General Availability = ADAPTION, RESULTS, CEL
 Variable Description = Velocity
 Variable Scope = PHASE
END
```

The following information can be identified:

- The name appearing immediately after `VARIABLE:` (in this case `vel`) is the CFX-Solver name or short name for the variable.
- For scalar quantities (`Tensor Type = SCALAR`), the name appearing after `VARIABLE:` is the name that must be used in any CEL expressions. The `MMS Name` and `Long Name` are not valid names to use for this purpose.
- For vector and tensor quantities (`Tensor Type = VECTOR` and `Tensor Type = SYMTEN2`), the `Component Short Names` (`u`, `v` and `w` in this case) are names that must be used in any CEL expressions.
- The `User Level` setting controls when the variable is seen. Variables of `User Level = 1` will appear in drop-down variable selection menus in CFD-Post. Variables with `User Level = 2` or `User Level = 3` will only appear in the full list of variables.
- The `Output to Postprocessor` setting controls if the variable is written to the results file for use in CFD-Post. A variable must be involved in the simulation before is has the potential to be written out.
- The `General Availability` field determines when a variable can be used. Only variables whose `General Availability` includes `CEL` can be used in CEL expressions.
- The names of all `VARIABLE`, `EQUATION DEFINITION` and `FUNCTION` objects are reserved names. These names should not be used as the name for any CEL expressions, Additional Variables, user routines, or user functions, etc.

The end of the `VARIABLES` file contains a section listing call back definitions. Using call backs which are not documented, or using them on locations other than those documented will produce invalid results. For details, see Quantitative Function List (p. 142) in the ANSYS CFX Reference Guide.

RULES File

The `RULES` file contains information about which models and parameters are valid options. This includes `SINGLETON`, `OBJECT` and `PARAMETER` items.

RULES

The first items in the `RULES` file list the top level `SINGLETON` objects. All other items are children of these optional top level objects.

- `RULES`: This could be used to modify the list of allowed options. This should always be done locally.

- LIBRARY: Includes libraries for materials, reactions, CEL and user routines. Add new materials and reactions using a local library.
- FLOW: Contains the current problem definition.
- USER: Where user parameters may be stored for later retrieval through User Fortran.
- COMMAND FILE: Contains the command file version number.
- EXECUTION CONTROL: Contains information about the parallel setup and other settings controlled in the CFX-Solver Manager.

Details about models that are not available can also be found through the user interface. For example, the SINGLETON: ADVECTION SCHEME item includes the QUICK scheme.

SINGLETON

A SINGLETON object is permitted to appear once as the child of a parent object. For this reason SINGLETON objects do not have names associated with them, for example:

```
LIBRARY :
 CEL :
  EXPRESSIONS :
```

Each of these three items is a SINGLETON object. The LIBRARY SINGLETON may only have one CEL SINGLETON, and the CEL SINGLETON may only have one EXPRESSIONS SINGLETON. Parameters and child definitions do not need to be grouped for a SINGLETON into a single location; this is done automatically by the CFX-Solver. For example, the following is valid:

```
LIBRARY :
 CEL :
  EXPRESSIONS :
   myexp1 = 1 [ m ]
  END
 END
END
...
LIBRARY :
 CEL :
  EXPRESSIONS :
   myexp2 = 2 [ m ]
  END
 END
END
```

OBJECT

An OBJECT is similar to a SINGLETON except that more than one can appear as the child of a parent object. For this reason each OBJECT must have a name, for example:

```
FLOW :
 DOMAIN : domain1
  ...
 END
 DOMAIN : domain2
  ...
 END
END
```

Two OBJECT items of type DOMAIN have been defined with names domain1 and domain2. These are children of the FLOW SINGLETON.

PARAMETER

A PARAMETER consists of a name, followed by the "=" character, followed by a value. Many parameters in the RULES file cannot be set in a CCL file and therefore cannot be altered.

- The Parameter Type tells you the type of values the parameter is allowed to take. It could be Real, Real List, Integer, Integer List, String or String List. When a list is valid, items should be comma separated.
- Some parameters contain an Allowed String List. This contains valid strings that can be used for that parameter.
- Each PARAMETER that you can set has a Dependency List which lists the variables the PARAMETER can depend upon. XYZ refers to the x, y and z coordinates. XYZT refers to the x, y and z coordinates and time. The CCL file can be edited based on the dependencies listed here.
- If a PARAMETER contains the item Dynamic Reread Item = Yes, then it can be modified while a run is in progress. Some SINGLETON objects also contain this item. For details, see Edit Run In Progress Command (p. 105).

Chapter 6. File Menu

This chapter describes the commands available from the CFX-Solver **File** menu:

Define Run Command

Running the CFX-Solver involves passing it the geometry, models, boundary conditions, and start-up information that it needs to calculate a solution to your CFD problem. The **Define Run** form is where you specify the information which is passed to the CFX-Solver. Some runs of the CFX-Solver require you to specify only the name of the CFX-Solver input file; others also require the name of an initial values file. Extra tabs become available when **Show Advanced Controls** is selected. For details, see The Define Run Dialog Box (p. 7).

Monitor Run in Progress Command

Used when CFX-Solver Manager has been closed and a run currently underway in the Solver needs to be displayed.

1. Select **File** > **Monitor Run in Progress**.

   The **Select a Run Directory (.dir)** dialog box is displayed.

2. Browse to the directory containing the current run.
3. Select the current run.
4. Click **OK**.

   Data up to the current timestep is loaded.

Monitor Finished Run Command

Used to view the residual plots of a finished run.

1. Select **File** > **Monitor Finished Run**.

   The **Monitor Finished Run** dialog box is displayed.

2. Under **File Type**, select the type of files to view.
3. Browse to the directory containing the finished run.
4. Select the run to view.

   If required, select a different file.

5. Click **OK**.

   The data is loaded.

Close Command

Closes all windows related to the current run. Any other runs that were open are not affected, and the last open run prior to the current run is displayed. If the Solver was in progress on the current run, it continues to operate in the background. The run can be re-monitored; see Monitor Run in Progress Command (p. 85) or Monitor Finished Run Command (p. 85), as appropriate.

Quit Command

Exits CFX-Solver Manager. Closing CFX-Solver Manager does not stop CFX-Solver jobs that are currently running. CFX-Solver Manager can be re-opened to take control of these jobs again simply by opening CFX-Solver Manager and selecting **Monitor Run in Progress**. For details, see Monitor Run in Progress Command (p. 85).

1. Select **File** > **Quit**.
2. Click **Quit**.

Chapter 7. Setting CFX-Solver Manager Options

The **Edit** > **Options** dialog enables you to set various general preferences. Settings are retained per user.

1. Select **Edit** > **Options**.

   The **Options** dialog box appears.

2. Set options as required. If desired, select **CFX Defaults** to use *all* of the default settings.

   If you are using ANSYS Workbench and want to use its default settings, select **Workbench Defaults**.

   For descriptions of the available options, see:

   - CFX-Solver Manager Options (p. 87)
   - Common Options (p. 88)

3. Click **OK**.

CFX-Solver Manager Options

When the **Options** dialog box appears, CFX-Solver Manager options can be set under **SolverManager**.

Default Layout Mode

You may specify a default layout mode to specify what mode monitors are presented when starting a new run. Select from the following:

- `Multiple Windows`
- `Tabbed`

Show Original Variable Names

If selected, the variable names will be shown with their original names. By default, this option is not selected and is generally left at that default setting.

Don't write Backup file on Edit Run In Progress

By default, a backup file is written. For details, see Edit Run In Progress Command (p. 105).

Default License Mode

The **Default License Mode** setting is applicable when using CFX in ANSYS Workbench when a license key supporting preprocessing, postprocessing, and solving is available. The options are:

- `Current WB License`

  CFX-Solver uses the license selected by ANSYS Workbench. The latter selects licenses according to the settings found under **Tools** > **Options** > **Licensing** > **License Management**. For details, see the ANSYS Workbench documentation.

  When running the CFX-Solver using this method, the license key capable of preprocessing, postprocessing, and solving is used for the solver, and becomes unavailable to provide access to meshing, pre-processing, or post-processing during that time.

- `Separate License`

  CFX-Solver checks out a separate license, leaving the ANSYS Workbench selected license key available so that you can use the latter to do other work while the solver is running.

Save Workspace to Results File

By default, this option is selected, enabling the CFX-Solver Manager workspace state to be saved automatically into the results file for the run. This means that if you subsequently re-monitor the run, any custom plots or plot settings that you used for the original run will be preserved.

Note that with this option selected, opening any ANSYS CFX results file in Release 11.0, or later, of the CFX-Solver Manager will modify the results file in such a way that it becomes incompatible with Release 10.0, or earlier, of the CFX-Solver Manager. In general, compatibility cannot be guaranteed whenever an ANSYS CFX results file from a particular release is opened in a CFX-Solver Manager from a later release. The results file will still be functional for the CFX-Solver from the earlier release, but you will not be able to monitor such a run using the CFX-Solver Manager from the earlier release. You should, therefore, disable **Save Workspace to Results File** when switching back and forth between different releases of the CFX-Solver Manager. You should also disable this option if you want to retain an unmodified results file.

Monitor

Multiple options exist that can be monitored as required.

- The visibility for each type of residual can be toggled on or off. Settings take effect the next time a run is started, or the next time a results file is viewed.
- If visibility is disabled for all residuals in a plot monitor, the monitor will not be created.
- For details, see Customizing CFX-Solver Manager (p. 5).

Monitor options are available to specify global residual display preferences. Settings chosen on the form apply for all future solver runs and override default display settings for monitors. Monitors can still be created using the **Workspace** menu. For details, see New Monitor Command (p. 97).

Common Options

Auto Save

Select the time between automatic saves.

To turn off automatic saves, set **Auto Save** to `Never`.

> **Note**
>
> This option affects more than one CFX product.

Temporary directory

To set a temporary directory, click *Browse*  to find a convenient directory where the autosave feature will save state files.

Appearance

The appearance of the GUI can be controlled from the **Appearance** options. The default GUI style will be set to that of your machine. For example, on Windows, the GUI has a Windows look to it. If, for example, a Motif appearance to the GUI is preferred, select to use this instead of the Windows style.

1. Under **GUI Style**, select the user interface style to use.
2. For **Font** and **Formatted Font**, specify the fonts to use in the application.

> **Note**
>
> It is important not to set the font size too high (over 24 pt. is not recommended) or the dialog boxes may become difficult to read. Setting the font size too small may cause some portions of the text to not be visible on monitors set at low resolutions. It is also important not to set the font to a family such as Webdings, Wingdings, Symbols, or similar type faces, or the dialog boxes become illegible.

> **Note**
>
> **Formatted Font** affects only the font used in CFX-Solver Manager for the out file display.

Viewer Setup

1. Select **Double Buffering** to use two color buffers for improved visualization.

   For details, see Double Buffering (p. 89).

2. Select or clear **Unlimited Zoom**.

   For details, see Unlimited Zoom (p. 89).

Double Buffering

Double Buffering is a feature supported by most OpenGL implementations. It provides two complete color buffers that swap between each other to animate graphics smoothly. If your implementation of OpenGL does not support double buffering, you can clear this check box.

Unlimited Zoom

By default, zoom is restricted to prevent graphics problems related to depth sorting. Selecting **Unlimited Zoom** allows an unrestricted zoom.

Mouse Mapping

The mouse-mapping options allow you to assign viewer actions to mouse clicks and keyboard/mouse combinations. These options are available when running in standalone mode. To adjust or view the mouse mapping options, select **Edit** > **Options**, then **Viewer Setup** > **Mouse Mapping**. For details, see Mouse Button Mapping (p. 49).

Units

1. Under **System**, select the unit system to use. Unit systems are sets of quantity types for mass, length, time, and so on.

   The options under **System** include `SI`, `CGS`, `English Engineering`, `British Technical`, `US Customary`, `US Engineering`, or `Custom`. Only `Custom` enables you to redefine a quantity type (for example, to use inches for the dimensions in a file that otherwise used SI units).

   The most common quantity types appear on the main **Options** dialog; to see *all* quantity types, click **More Units**.

2. Select or clear **Always convert units to Preferred Units**.

   If **Always convert units to Preferred Units** is selected, the units of entered quantities are immediately converted to those set on this dialog.

   For example, if you have set **Velocity** to `[m s^-1]` on this dialog to make that the preferred velocity unit, and elsewhere you enter `20 [mile hr^-1]` for a velocity quantity, the entered value is immediately converted and displayed as `8.94078 [m s^-1]`.

Chapter 8. Workspace Menu

The **Workspace** menu controls layout, plots, and text windows, which are visible in the viewer. With the **Workspace** menu you can:

- Back up, restart, or stop the current run
- Change the properties for the current workspace
- Create new monitors for simulations
- Switch between multiple windows and tabbed GUI layout (using the **Toggle Layout** menu option)
- Switch between viewing RMS and Maximum residual values at any time during or after a run.

This chapter describes:

Workspace Properties Command

By default, appropriate monitors are automatically created by CFX-Solver Manager for the particular type of simulation you are running. The type and/or number of plots that appear can be changed by using **Workspace Properties**. Not all of the default plots relevant to the solution are displayed: you can choose them in the **Monitors** menu. For details, see *Monitors Menu* (p. 107).

When loading a results file or starting a solver run, CFX-Solver Manager checks the type of run before creating the correct plots. For example, a volume fraction plot will be created for multiphase simulations.

Workspace properties are controlled from:

General Settings Tab

This tab displays the following workspace information that cannot be edited.

- **Out File** displays the path and name of the CFX output file for the run displayed in CFX-Solver Manager.
- **Directory** displays the path and name of the directory for the run displayed in CFX-Solver Manager.

The locations can be set in the **Working Directory** option when defining the run. For additional information, see Define Run Command (p. 85) and Run Definition Tab (p. 8). Information on selecting different runs is available in Workspace Selector (p. 2).

Monitors Tab

The **Monitors** tab allow customization of the CFX-Solver Manager viewer area and text windows. The following types of monitors are available:

- Plot Monitor (p. 92)
- Residual Monitor (p. 92)
- Text Monitor (p. 92)

Plot Monitor

Plot monitors show the values of expressions versus timestep.

Residual Monitor

Residual monitors show the values of residuals for equation variables versus timestep.

Text Monitor

Text monitors show the contents of text files that are updated as the run proceeds.

Filter Selector

The Filter selector is a drop down list that shows the monitor types available in CFX-Solver Manager. Use this selector to control the monitors displayed under the **Monitors** tab by selecting `All`, `Plot Monitor`, `Residual Monitor` or `Text Monitor`. The `All` setting displays a full list of available monitors.

Creating Monitors

Monitors can be created within **Workspace Properties**. However, they can also be created directly from the **Workspace** menu by selecting **New Monitor**.

1. Select **Workspace** > **Workspace Properties**.
2. Select **Monitors**.
3. Click *New*  .

   The **New Monitor** dialog box is displayed.
4. Under **Name**, type the name of the new monitor.
5. Under **Type**, select `Plot Monitor`, `Residual Monitor` or `Text Monitor`.
6. Click **OK**.

   The **Monitor Properties** dialog box is displayed.
7. Configure the monitor as required.
   - General Settings Tab (p. 93)
   - Range Settings Tab (p. 93)
   - Plot Lines Tab (p. 94)

Modifying Monitors

1. Select **Workspace** > **Workspace Properties**.
2. Select **Monitors**.
3. Select the monitor to modify.
4. Click *Edit*  .

   The **Monitor Properties** dialog box is displayed.

5. Configure the monitor as required.
   - General Settings Tab (p. 93)
   - Range Settings Tab (p. 93)
   - Plot Lines Tab (p. 94)

Deleting Monitors

1. Select **Workspace** > **Workspace Properties**.
2. Select **Monitors**.
3. Select the monitor to delete.
4. Click *Delete*  .

Monitor Properties

Monitor properties differ depending on the type of monitor. Up to three tabs are available to configure monitor properties.

- General Settings Tab (p. 93)
- Range Settings Tab (p. 93)
- Plot Lines Tab (p. 94)

General Settings Tab

1. Under **Window Label**, type the name to display for the monitor.
2. If working with a text monitor:
   1. Under **Text File Name**, click *Browse*  and select a file containing the definition for the text monitor.

      This can be any .out file.
   2. Select or clear **Disable this Monitor**.

      If selected, the monitor is disabled.
3. If working with a plot monitor or a residual monitor:
   1. Under **Background Color**, click *Color Selector*  and set the background color.
   2. Select or clear **Display Legend**.

      If selected, the legend is displayed in the monitor.
   3. Under **Grid Mode** select `Both`, `X`, `Y` or `None`.

      This determines if grid lines appear along the X or Y axis, on both or not at all.
   4. Under **Grid Color**, click *Color Selector*  and set the grid color.
4. Select or clear **Visibility**.

   If selected, the monitor is displayed.

Range Settings Tab

1. Under **Timestep Range Option**, select `Display All`, `Most Recent`, `Fixed` or `This Run Only`.
   - `Display All` displays values for every iteration. If viewing a restarted run, results from the previous run are also visible.
   - `Most Recent` displays the current iteration and a number of previous iterations.

- `Fixed` displays a beginning and end iteration which is always displayed, regardless of the current iteration number.
- `This Run Only` displays the range for the current run. If the run is a restart, previous runs are not included in the range.

2. Under **Variable Axis**, select or clear **Use Logarithmic Scale**.

   If selected, **Set Manual Scale (Log)** is automatically selected.

3. Under **Set Manual Scale (Log)** or **Set Manual Scale (Linear)** set the **Upper Bound** and **Lower Bound** values for the variable axis.
4. Complete the timestep configuration based on the **Timestep Range Option** selected.
   - **Most Recent** requires a value for **Timestep Window Size**.
   - **Fixed** requires values for **First Timestep** and **Last Timestep**.

Plot Lines Tab

Variables which are available to plot can come from a variety of sources. Select the source by setting **Variable Set** appropriately. Available settings are:

- **CFX Solver**: This option will be available for all CFX runs, and allows the specification of all variables relating to the CFX run.
- **ANSYS Field Solver**: This option is only available if an ANSYS Multi-field Run is being performed with ANSYS running a non-linear simulation, and allows the selection of ANSYS convergence variables.
- **ANSYS Interface Loads**: This option is only available if an ANSYS Multi-field Run is being performed, and allows the selection of the variables relating to the data exchanged between the ANSYS and CFX Solvers.

Plot lines can be displayed as required:

1. Expand a plot line variable by clicking + next to it.

   Categories are displayed.

2. Expand a category by clicking + next to it.
3. Select or clear specific plot line variables.
4. If working with a Residual Monitor, under **Residual Mode**, select `RMS` or `MAX`.

   Residual Mode acts as a visibility filter for variables of Residual type selected by default. If a variable of type Residual is not selected by default it is plotted even if its type does not match the Residual Mode setting.

5. Click **Apply**.

CFX Variables

While not all variable types are available at any given time, the following is a complete list of all variable types available for monitoring a CFX run.

- ACCUMULATED TIMESTEP (p. 95)
- BALANCES (p. 95)
- FLOW (p. 95)
- FORCE (p. 95)
- NEG ACCUMULATION (p. 95)
- RESIDUAL (p. 95)
- SOURCE (p. 95)
- TIMESTEP (p. 95)
- USER (p. 95)

Further details on the output of the solver are available. For details, see *CFX-Solver Files* (p. 25).

ACCUMULATED TIMESTEP

The accumulated timestep values are plotted.

BALANCES

These are the normalized sum of the flows (that is, the % imbalance) for a given equation on a particular domain. The absolute flow is normalized by the maximum flow, calculated by looking at flows on all domains for that particular equation.

FLOW

The flows listed in the CFX-Solver Manager are the absolute amounts of a variable transported through a boundary condition. For example, the flow for the continuity (P-Mass) equation is the mass flow of a particular phase through the boundary condition. The flow for the energy (H-Energy) equation is the energy flow per unit time through the boundary condition.

FORCE

For details, see Calculated Wall Forces and Moments (p. 40).

NEG ACCUMULATION

Negative accumulation is the transient term contribution to the balance equation. For details, see BALANCES (p. 95).

RESIDUAL

For details, see *Residual Plotting* (p. 71).

SOURCE

Sources are the amount of a variable created or consumed in a particular three dimensional region of a domain. The region might be user defined, if user defined source terms were set up, or the region might be the entire domain. Source values which are defined for the entire domain are automatically computed by the solver, and vary depending on the models which are being used. For example, automatically computed source values will appear in the turbulent kinetic energy equation which represent the production and dissipation of turbulence.

TIMESTEP

Under the TIMESTEP heading, the only variable available is `Accumulated Time Step`. CFX-Solver Manager uses this variable to plot the x-axis of ANSYS Field Solver and ANSYS Interface Loads plots. It represents the cumulative iteration number in the following way:

- For the ANSYS Field Solver plot, the cumulative iteration number counts the number of iterations which the ANSYS solver has performed in order to converge. Several ANSYS iterations will be performed for each timestep or coupling step, depending on how quickly ANSYS converges.
- For the ANSYS Interface Loads plot, the cumulative iteration number counts the number of stagger iterations performed by the ANSYS Multi-field solver.

USER

If monitor points have been created, a `USER` category is available. This can be expanded to select the monitor points to plot. For details, see Monitor Tab (p. 157) in the ANSYS CFX-Pre User's Guide.

Configuring Plot Lines

Plot line variables can be configured as required.

1. Expand a plot line variable by clicking + next to it.

   Categories are displayed.

2. Expand a category by clicking + next to it.
3. Select or clear specific plot line variables.
4. If working with a Residual Monitor, under **Residual Mode**, select `RMS` or `MAX`.

Residual Mode acts as a visibility filter for variables of `Residual` type selected by default. If a variable of type `Residual` is not selected by default it is plotted even if its type does not match the **Residual Mode** setting.

5. Click **Apply**.

ANSYS GST and NLH Variables

The variables available for plotting when monitoring the contents of ANSYS GST and NLH files are mostly common, and are described together.

ANSYS

The variables that are typically present in a GST or NLH files under the ANSYS heading are listed in the table below together with a brief description (for the context of an ANSYS Multi-field MFX simulation). Other variables may be present depending on the solution physics and the specified convergence targets. For full details, see the ANSYS Product documentation.

Variable Name	Description
Bisection	For details, refer to the ANSYS documentation in “Timestep Bisection”.
Line Search Parameter	For details, refer to the ANSYS documentation in “Line Search”.
Load Step	In the context of an ANSYS Multi-field run, `Load Step` is the same as a multi-field timestep or `Coupling Step`.
Max DOF Incr	The largest change detected in the degree of freedom of interest over the last two equilibrium iterations
Sub-step	The ANSYS field solver may be set to use a smaller timestep than the multi-field timestep. This is known as sub-cycling. In this case, the multi-field timestep is broken into a series of sub-steps.
Time	Multi-field time (current simulation time for a transient case)
Time Incr	Current time increment (may be variable if adaptive timestepping is used)
<variable> CRIT, <variable> L2, <variable> L1, <variable> INF	Only in the GST file. For details, see ANSYS Field Solver Plots (p. 72).
UX, UY, UZ, FX, FY, FZ, TEMP, HFLU	Only in the NLH file. For details, see ANSYS Interface Loads Plots (p. 73).

Plotting by a Specific Variable

While the CFX-Solver is running, the default setting is to plot the data as a function of `Accumulated Time Step`. However, you may choose to view the data as a function of any available variable.

For example, you may choose to plot an important monitored quantity as a function of residual level. The existing residual plot provides information on the exact convergence level, while this plotting feature gives another way to determine how far you need to converge the solution, by combining residual information with monitored quantities. For example, if you were to plot `Mass Averaged Outlet Total Pressure` vs. `Mass Residual` you would see how quickly this important quantity stabilizes as a function of the residual convergence level. You may then set a residual target for future runs that allows the CFX-Solver to stop when the established residual levels are sufficient for the "important monitored quantity".

You can change the variable that is plotted on the X-axis for any specific plot as follows:

1. Right-click the plot you wish to modify (see Convergence History Plots (p. 2)) and select **Monitor Properties** from the shortcut menu.

2. On the **Range Settings** tab, set **Plot Data By** to `Specified Variable` and select the variable you wish to use as the X-axis variable.

Global Plot Settings Tab

Plotting of coefficient loop/cloop data is turned off by default.

For transient runs, this selects or clears plotting of cloop data for each inner loop. This is in addition to plotting data for each outer loop/timestep. For details, see Transient Residual Plotting (p. 72).

> **Note**
>
> As turning on the cloop setting results in a three to seven times increase in the size of the monitor data file, it affects the performance of the CFX-Solver Manager.

1. Select **Workspace** > **Workspace Properties** > **Global Plot Settings**.
2. Select or clear **Plot Coefficient Loop Data**.
3. Under **Plot Data By** select `Time Step` or `Simulation Time`.

   Variables are plotted against accumulated `Time Step` (the default) or against `Simulation Time`. The latter is useful when using non-uniform timesteps.

New Monitor Command

This menu allows a shortcut to the creation of new monitors. However, if other properties associated with a workspace need to be defined, full **Workspace Properties** should be viewed. Monitors can then be created as required.

- Workspace Properties Command (p. 91)
- Monitors Tab (p. 92)

Stop Current Run Command

To stop the CFX-Solver as soon as possible, use one of the following methods:

- Select **Workspace** > **Stop Current Run**.
- Click *Stop Current Run*  .

If running CFX-Solver from the command line, use the command: `cfx5stop`. For an example of the `cfx5stop` command, see Command-Line Samples (p. 118).

When CFX-Solver stops, the run is marked as finished, and a message appears. This message names the run and specifies it has terminated at your request. Additional information about the run is also listed. Once manually terminated, a run can be manipulated in the same way that a completed run .

Runs Using Mesh Adaption

If a stop current run command is issued for a run that uses mesh adaption, then only the current CFX-Solver run is terminated; the overall simulation, which includes equation solution and mesh adaption, is not. The CFX-Solver is automatically restarted after executing the next adaption step (if any remain), and the simulation continues.

Runs Using Remeshing

If a stop current run command is issued for a run that uses remeshing, then the overall simulation is terminated.

Runs Using External Solver Couplings

If a stop current run command is issued for a run that uses external solver couplings (for example, ANSYS Multi-field), then the overall simulation is. For ANSYS Multi-field couplings, termination occurs at the end of the current **Load Step**, for details see the *Coupled Field Analysis Guide* in the Mechanical APDL user documentation.

In particular, steady state analyses will run until completion, and transient analyses will run until the end of the current time step.

Multi-Configuration Runs

If a stop run command is issued in a workspace for a specific configuration, then only the current CFX-Solver run for that configuration will be terminated; the overall simulation is not. All remaining configurations that are defined for the simulation are subsequently run (unless a stop run command is also issued for them). If a stop run command is issued in the simulation-level workspace, which is labeled according to the name of the multi-configuration CFX-Solver Input file (`*  .mdef`), then the overall simulation is terminated. The CFX-Solver run for the configuration currently being executed is terminated as soon as possible, and no additional configurations are executed.

Restart Current Run Command

You can restart a run that is finished or stopped. Restarting a run starts the run with the same settings as the previous run, including Parallel settings.

There are numerous ways a run can be set up to restart; these are described in Restarting a Run (p. 16). To perform the restart, either:

- Select **Workspace** > **Restart Current Run**.
- Click *Restart Current Run*  .

Backup Run Command

Backing up a run creates a backup file of the results at the end of the timestep that is currently calculating. This file contains sufficient information for restarting a run for visualization. You should generate backup files if a solution may be diverging and an intermediate solution needs to be retained.

To back up a run, either:

- Select **Workspace** > **Backup Run**.
- Click *Backup Run*  .

The backup file is stored in a subdirectory within the working directory. This subdirectory is given the same name as the current run.

Arrange Command

Arranging the workspace deletes all monitors that are currently showing, regenerates them, and redisplays them by optimizing the display based on available screen space. This has no impact on the display if the layout type has been toggled to display multiple overlapping tabs. For details, see Toggle Layout Type Command (p. 98).

To arrange the workspace, either:

- Select **Workspace** > **Arrange**.
- Click *Arrange Workspace*  .

Toggle Layout Type Command

Two layout modes exist. One is set up with multiple overlapping tabs used to switch between monitors and the other displays each monitor in its own window. These may be toggled as required.

To toggle the workspace layout, either:

- Select **Workspace** > **Toggle Layout Types**.
- Click *Toggle Layout Types*  .

Load Layout Command

Layouts can be loaded as required. Before loading a layout, there must be layouts that have been saved. For details, see Save Layout Command (p. 99).

To load a previously saved layout:

1. Select **Workspace** > **Load Layout** or click *Load Layout*  .

   The **Layout File** dialog box is displayed.

2. Select the location containing the file to load.
3. Select the file to load.
4. Click **Open**.

   As the new workspace replaces the existing one CFX-Solver Manager requires confirmation that the new workspace should overwrite the existing one.

5. Click **Yes**.

   The new layout is loaded.

Save Layout Command

Once a layout has been configured to display preferred settings views, it can be saved. Once saved, layouts can be loaded as required. For details, see Load Layout Command (p. 99).

This is useful when carrying out different runs for the same problem. For example, there may be a layout with preferred settings after changing a boundary condition value. Another layout may be preferred for viewing a turbulence model. By saving and loading a layout, it is simple to switch between these views.

The saved layout fully restores settings only for problems with the same domain and boundary names as the problem that was selected in the when the layout was saved. If the saved layout is loaded when the current problem has different domain and/or boundary names, variables to plot must be reselected. For details, see Duplicating a Plot Monitor (p. 99).

1. Configure the current layout to the appearance to save.

   That is, display various monitors, position them, set up the layout type and so on.

2. Select **Workspace** > **Save Layout**.

   The **Layout File** dialog box is displayed.

3. If required, set the path location to a different directory.
4. Under **File name**, type the name of the file to save.
5. Click **Save**.

   If the name already exists, a warning dialog box is displayed.

   - Overwrite replaces the old file with the new document.
   - Re-select returns to the Layout File dialog box.
   - Cancel closes the open dialog boxes.

Duplicating a Plot Monitor

Rather than creating plot monitors manually, an existing one can be duplicated and modified.

The easiest way to create your own plot monitor is to copy the definition of an existing monitor and edit it to include the new variables. The default monitor files (`.mst`) can be found in the `<CFXROOT>/etc/CFX/` directory. For example, if you want to view turbulence data on the same plot as mass and momentum you could enter the following:

```
^RESIDUAL,RMS,.*-Mom;^RESIDUAL,RMS,.*-Diss.K; \
^RESIDUAL,RMS,.*-TurbKE;
```

As stated previously, the string for which a search is made can be found from the variable lists in the **Plot Lines** panel of the **Monitors** form. For details, see Plot Lines Tab (p. 94).

View RMS Residuals Command

To display the RMS values of the residuals, select **Workspace** > **View RMS Residuals**.

For more information about residuals, see *Residual Plotting* (p. 71).

View MAX Residuals Command

To display the MAX values of the residuals, select **Workspace** > **View Max Residuals**.

For more information about residuals, see *Residual Plotting* (p. 71).

Reset to Default Workspace Command

Resets the default workspace to the state it was in immediately after the run was started, or after a finished run was loaded. This can be useful when retrieving plots that have been accidentally deleted, or when reloading the original plots after a change to their definition.

> **Note**
>
> Resetting the default workspace deletes any custom monitors.

1. Select **Workspace** > **Reset to Default Workspace**.
2. Click **Yes**.

Close Command

For information on the Close command, see Close Command (p. 85).

Chapter 9. Tools Menu

The **Tools** menu controls layout, plots and text windows which are visible in the viewer. The current run can also be backed up, restarted or stopped.

Properties for the current workspace can be changed, and new monitors created for simulations.

This chapter describes:

Edit CFX-Solver File Command

The command file section of a CFX-Solver input file can be modified. This allows modifications to a CFX-Solver input file without the need to use CFX-Pre. The most useful application of this is in the modification of a simulation when re-running the simulation may be too time consuming.

When a CFX-Solver input file is selected for modification, the Command File Editor is launched. For details, see *Editing CFX-Solver Input Files* (p. 75).

1. Select **Tools** > **Edit CFX-Solver File**.
2. Browse to the directory containing the CFX-Solver input file to edit.
3. Select the CFX-Solver input file.
4. Click **Open**.

   The Command File Editor is launched.

Export Command

If using tools other than CFD-Post for post-processing, data needs to be exported to a results file in a supported format. The CFX Export utility can also be run from the command line.

- *File Export Utility* (p. 131)
- Running cfx5export from the Command Line (p. 143)

There must be a results file to reference before exporting.

1. Select **Tools** > **Export**.

   The **Export** dialog box is displayed.

2. Under **Results File**, click *Browse*  and select a results file for export.

3. If required, under **Export File**, click *Browse*  and modify the default output path and name.

4. Under **Domain Name**, select the domain to export.

   Where multiple domains exist, select the domain(s) to export.

5. Under **Export Type** select `CGNS`, `MSC.Patran`, `FIELDVIEW`, `EnSight` or `Custom User Export`.
   - CGNS (p. 133)
   - MSC.Patran (p. 135)

- FIELDVIEW (p. 138)
- EnSight (p. 140)
- Custom User Export (p. 142)

6. Under **Export Options**, configure options as required.

   The options are dependent on the Export Type. For details, see *File Export Utility* (p. 131).

7. Click **Export**.

   Once completed, a message is displayed. Click **OK** to close it.

Export to ANSYS MultiField Command

This option is used to produce files for use in Fluid Structure Interaction cases. For details, see Export to ANSYS Multi-field solver Dialog (p. 131).

1. Select **Tools** > **Export to ANSYS MultiField**.

   The **Export to ANSYS MultiField Solver** dialog box is displayed.

2. Under **Results File**, click *Browse*  and select a results file for export.

3. If required, under **Export File**, click *Browse*  and modify the default output path and name.

4. Under **Domain Name**, select the domain to export.

   Where multiple domains exist, select the domain(s) to export.

5. If required, under **Boundary**, select the boundaries to export.

6. Under **Export Options**, configure options as required.

   The options are dependent on the export type. For details, see Export to ANSYS Multi-field solver Dialog (p. 131).

7. Click **Export**.

   Once completed a message is displayed. Click **OK** to close it.

Interpolate Command

ANSYS CFX allows the values from one results file to be interpolated onto a CFX-Solver input file containing another mesh.

The major benefit of interpolation is the ability to use the solution from a simple model to provide initial conditions for another, perhaps more complex model (thus increasing the likelihood of converging the complex model simulation) or to continue a run with a different mesh or other settings.

Interpolation can be used with modified geometry or boundary conditions. Interpolation can also be used to interpolate the solutions from a model with different mesh topology. For example, the initial guess for a problem having one domain can be interpolated from one or more results files having a solution that spans multiple domains. When the shape of the model has changed and the initial values files do not fully overlap with the new mesh, the CFX-Interpolator extrapolates values for the points in the new mesh that lie outside the Initial Values File(s), based on the interpolated values on the mapped nodes. See Using the CFX-Interpolator (p. 86) in ANSYS CFX-Solver Modeling Guide for more details on how the CFX-Interpolator works.

The CFX-Interpolator is most commonly invoked through settings on the **Run Definition** tab on the **Configuration** or **Execution Control** details view in CFX-Pre, or, the **Define Run** dialog of the CFX-Solver Manager. For details, see Run Definition Tab (p. 225) in ANSYS CFX-Pre User's Guide, Run Definition Tab (p. 216) in ANSYS CFX-Pre User's Guide and Run Definition Tab (p. 8), respectively.

However, if you wish to manually use the interpolator to write variables into a specific Solver Input File (not invoked from the **Run Definition** tab) then you can choose **Tools** > **Interpolate Results** from the CFX-Solver Manager. In this case the specified **Mesh File** is used as the target file, and will be modified by the interpolation process. The

text output is written into the CFX-Solver Manager's Interpolation dialog. You can then run the resulting target file in the CFX-Solver, using the variables written into the target file by the CFX-Interpolator as the initial conditions for the run.

To manually interpolate the results from a source file to a target file, you can use the **Interpolate Results** command as follows:

1. Select **Tools** > **Interpolate Results**.

   The **Interpolation** dialog box is displayed.

2. Select the **Interpolate** method.

3. Set **Results File** to the name of the source file. You can click *Browse*  to select the file using a browser.

4. Set **Mesh File** to the name of the target file. You can click *Browse*  to select the target file using a browser.

   You can interpolate onto a CFX-Solver Input file.

5. Under **Executable Settings**, select or clear **Double Precision**.

   For details, see Double-Precision Executables (p. 123).

6. If required, under **Interpolator Memory**, adjust the memory configuration. For details, see Configuring Memory for the CFX-Solver (p. 13).

7. Click **Interpolate**.

   The output window on the right displays details of the interpolation process.

   Once completed, a message is displayed. Click **OK** to close it.

You can also run the interpolator from the command line. For details see Using the Command Line to Interpolate Results (p. 104).

Note

- To save the results to a text file, right-click in the text output window and select **Save As**.
- Interpolating results from a source file to a target file which already contains solution data is not recommended. See Interpolating Onto a Solver Input File with Results Fields (p. 94) in ANSYS CFX-Solver Modeling Guide for details.
- Following the steps above, the interpolation is performed in Initial Guess mode. See Using the CFX-Interpolator (p. 86) in ANSYS CFX-Solver Modeling Guide for details.

To generate a comparison of two files, you can use the **Interpolate Results** command. The comparison is stored as new variables with the prefix `Difference`. These variables can be used in CFD-Post to determine regions where the solution has changed significantly. A comparison of two files can be generated as follows:

1. Select **Tools** > **Interpolate Results**.

   The **Interpolation** dialog box is displayed.

2. Select the **Calculate Differences** method.

3. Set **Original Results** to the name of the original results file. You can click *Browse*  to select the file using a browser.

4. Set **Modified Results** to the name of the modified results file. You can click *Browse*  to select the file using a browser.

5. Under **Executable Settings**, select or clear **Double Precision**.

   For details, see Double-Precision Executables (p. 123).

6. If required, under **Interpolator Memory**, adjust the memory configuration. For details, see Configuring Memory for the CFX-Solver (p. 13).

7. Click **Calculate Differences**.

The output window on the right displays details of the differencing process.

Once completed, a message is displayed. Click **OK** to close it.

> **Note**
>
> CFD-Post now also supports comparison of two results files without the use of the CFX-Interpolator. For details, see Case Comparison (p. 181) in ANSYS CFD-Post User's Guide.

Using the Command Line to Interpolate Results

In some instances, the `cfx5interp` script can be used from the command line to initiate an interpolation. This section presents a brief introduction to this script. For more information about this script, type `<CFXROOT>/cfx5interp -help` at the command prompt.

To use the `cfx5interp` script, enter a command line of the form:

```
cfx5interp -res <results file> -mesh <CFX-Solver input file> [<arguments>]
```

where:

- `<results file>` is the name of a results file that contains a solution.
- `<CFX-Solver input file>` is the name of a CFX-Solver input file that contains a mesh onto which the solution should be interpolated.
- `[<arguments>]` is an optional list of additional arguments.

`cfx5interp` rewrites the CFX-Solver input file with the interpolated form of the solution that was read from the results file. A command of this form would produce the same results as running invoking interpolation using **Tools** > **Interpolate Results** within the CFX-Solver Manager.

You can use the `-difference` argument to cause the interpolator to produce difference datasets for differences in results between two results files. The form of such a statement is:

```
cfx5interp -difference -from <res file 1> -dest <res file 2> [<arguments>]
```

where:

- `<res file 1>` is the name of a results file that contains the original (older) set of fields
- `<res file 2>` is the name of a results file that contains the newer set of fields
- `[<arguments>]` is an optional list of additional arguments

`cfx5interp` calculates the differences between an original set of fields and the newer set, and rewrites `<res file 2>` with the differencing information included.

The `cfx5interp` script can run two versions of the interpolator. The newer version (a solver-based interpolator) runs by default, and the older one (that was released with ANSYS CFX 10.0) can be invoked by using the `-interp-old` argument.

When the older interpolator is used, the `cfx5interp` script is capable of producing a text file of results for specific locations within the fluid domain. This is particularly useful if there is experimental data to validate.

To produce a results text file, first create a text file containing the particular vertex coordinates of interest, in the following format:

```
x(1) y(1) z(1)
x(2) y(2) z(2)
. . . .
x(n) y(n) z(n)
```

Once the vertex file is created, run the old interpolator using a command line of the form:


```
cfx5interp -vtx <vertex file> -res <results file> -interpolate-old
```

The old interpolator creates a file with a name of the form: `<vertex file>.inn`, where `nn` is chosen to make a unique file name. This is a text file that contains the coordinates that are specified in the vertex file, plus the results from the results file interpolated to the vertex locations.

If, in the vertex file, there are coordinates that lie outside of the solution grid, values of 0.0E0 will be assigned for all variables at those coordinates. In other words, results are not extrapolated to a vertex file.

> **Note**
>
> Some of the values obtained using the `cfx5interp` script may differ slightly from the values obtained using Data Export in CFD-Post. These minor discrepancies result from different methods of calculation. Discrepancies are more likely to occur at points which lie very close to the edge of the mesh elements or in regions of prism and hexahedral elements. Inconsistencies are likely to be more significant where gradients are large, particularly in the boundary layer.

Edit Run In Progress Command

You can edit the CCL definition of a CFX-Solver input file while the solver is running. The changes you make take effect when saved and the modified CCL is preprocessed for the flow solver. The modified CCL may take several iterations to be updated depending on system load, hardware and problem size.

Theses changes apply only to the run in progress, and do not affect the CFX-Solver input file that was used to begin the run (if one was used). Before the next outer-loop iteration begins, a backup file named `<iteration number>.bak` is written to the working directory. The backup file can be used to restart the run from the point at which the CCL definition was changed, if needed.

Only selected CCL parameters can be dynamically changed. A list of these parameters is in the RULES file. For details, see RULES File (p. 81).

The Command File Editor can also be used to make changes to the CCL contained in a CFX-Solver input file. Changes, when saved, affect the edited CFX-Solver input file only, not any run in progress. For details, see *Editing CFX-Solver Input Files* (p. 75).

To access the command file editor, select **Tools** > **Edit Run In Progress** or click *Edit Run In Progress*  .

> **Note**
>
> The **Edit Run in Progress** command is not enabled for simulations involving ANSYS Multi-field couplings. This constraint exists because of difficulties involved with updating the MFX settings in the ANSYS solver during the run.

CCL Propagation in Multi-Configuration Simulations

The **Edit Run in Progress** command is enabled for the configuration being run, rather than in the global (or simulation) level workspace. In addition to applying CCL changes to the running configuration, some or all of the changes are also automatically propagated to subsequent configuration runs. In particular:

- changes made to the `LIBRARY` section of the CCL (e.g. material properties, expressions, reactions, etc…) are propagated to all subsequent configurations, and
- changes made to the `FLOW` section of the CCL (e.g. analysis type, boundary conditions, solver and output controls, etc…) are propagated to all subsequent runs of the current configuration.

Edit Current Results File Command

Edit the results file in the current workspace (if available). This feature is available when a results file in the current workspace that has finished. For details, see *Editing CFX-Solver Input Files* (p. 75).

- Select **Tools** > **Edit Current Results File** or click  .

  The Command File Editor is launched and the current results opened.

Post-Process Results Command

Loads the CFD-Post post-processor. For details, see Overview of CFD-Post (p. 1) in the ANSYS CFD-Post User's Guide.

1. Select **Tools** > **Post-Process Results**.

   The **Start CFD-Post** dialog box is displayed.

2. Under **Results File**, click *Browse*  and select a results file to load into CFD-Post.
3. If you want to load two results files into CFD-Post together, check **Specify Additional Results File**, and select another results file to load.
4. Select or clear **Shut down Solver Manager**.

   If selected, ANSYS CFX-Solver Manager is shut down before CFD-Post is launched.

5. Click **OK**.

View Environment Command

Used to display a complete list of environment variables associated with the CFX-Solver Manager and their settings.

1. Select **Tools** > **View Environment**.

   The **Solver Manager Environment** dialog box is displayed.

2. Click **Save** to export the content to a text file.

Chapter 10. Monitors Menu

The **Monitors** menu sets the display options for plots of your simulation.

Each menu option has a submenu. This is used to specify display options for a given category. Selected options will display the related plot.

Residuals are combined by default in the CFX-Solver Manager. Residuals for each domain can be displayed as required by selecting the residuals by domain in the submenu for each residual type.

Chapter 11. Starting the CFX-Solver from the Command Line

The CFX-Solver is a separate module of CFX that has no graphical user interface. You can start CFX-Solver from the command line by executing the following command:

```
cfx5solve [options]
```

where [*options*] denotes the options applicable during a command-line run. This chapter discusses how to run CFX-Solver in a batch mode and describes the supported command-line options in the following sections:

You can also use CFX-Solver Manager to start CFX-Solver. The graphical user interface of the CFX-Solver Manager enables you to set various options, allows easier control of the solution process, and provides some visual details as the solution emerges. For details, see *CFX-Solver Manager Basics* (p. 1).

Command-Line Use

CFX-Solver Manager and CFX-Solver can be launched from a command line as follows:

- The basic command to start CFX-Solver Manager is `cfx5solve`.

  The more general form of the command is:

  ```
  cfx5solve [-interactive [-definition <file>]]
  [-display <display>] [-help]
  [-solver <executable>] [-verbose]
  ```

  where [] denotes a discretionary option, | separates mutually exclusive options, and <> denotes that substitution of a suitable value is required. All other options are keywords, some of which have a short form.

- The basic command to start the CFX-Solver using the CFX-Solver input file named `<file>` is:

  ```
  cfx5solve -def <file> [-help] [-initial <file>]
  [-double | -single]
  [-nosave|-save] [-name <name>]  [-size <factor>]
  [-solver <executable>] [-partition <number of partitions>]
  [-parallel] [-parfile <file>] [-serial] [-verbose]
  ```

  where [] denotes a discretionary option, | separates mutually exclusive options, and <> denotes that substitution of a suitable value is required. All other options are keywords, some of which have a short form.

How you invoke a command line depends on your operating system:

- On UNIX, you can run CFX-Solver Manager from a UNIX shell.
- On Windows, start a CFX Command Line from the CFX Launcher: **Tools** > **Command Line**. Alternatively, you can run CFX-Solver Manager from a DOS prompt. For details, see Command Line (p. 3) in the ANSYS CFX Reference Guide.

The `cfx5solve` command-line options are described in the next section.

Command-Line Options and Keywords for cfx5solve

The command-line options for `cfx5solve` are described below. To see command-line help, run `cfx5solve -help`.

Note

When running the solver from the command line using a CFX-Solver input file or CFX-Solver results file, any execution control CCL contained in the file takes precedence over the command-line options.

If an option is specified multiple times within the context of a specific configuration, then the last specification of the option takes precedence.

Command-Line Options	Alternative form	Usage
`-ansys-arguments <arguments>`		For an ANSYS Multi-field run, enables you to specify any additional options when starting the Mechanical application solver. The specified options are passed to the Mechanical application solver as command-line arguments. For details, see MultiField Tab (p. 9).
`-ansys-input <file>`		For an ANSYS Multi-field run, specifies the Mechanical application input file to use. For details, see MultiField Tab (p. 9).
`-ansys-input-is-complete`		For an ANSYS Multi-field run, this turns off the **Processing Mechanical input file** option. For details, see MultiField Tab (p. 9).
`-ansys-installation <directory>`		For an ANSYS Multi-field run, sets the ANSYS installation directory. This option is needed only if ANSYS is installed in a non-standard location. For details, see MultiField Tab (p. 9).
`-ansys-jobname <name>`		For an ANSYS Multi-field run, sets the jobname for the Mechanical application component of the simulation. The CFX-Solver Manager defaults this to `ANSYS`. On a restart, the jobname must be the same as for an initial run.
`-ansys-license <licensekey>`		For an ANSYS Multi-field run, sets the license that the Mechanical application solver should use. For details, see MultiField Tab (p. 9).
`-ansys-restart <file>`		For an ANSYS Multi-field run, tells CFX that the Mechanical application component of the simulation is a restart, and gives the database (`*.db` or `*.rdb`) from the previous Mechanical application run to be used for the restart. For details, see MultiField Tab (p. 9).
`-bak-elapsed-time <elapsed time frequency>`	`-baket <elapsed time frequency>`	Causes the flow solver to write a backup file every *<elapsed time frequency>* hours, minutes, seconds, et cetera. Elapsed time must be in quotes and have units in square brackets. For example: `-baket "10 [min]"` or `-baket "5 [hr]"`.
`-batch`		Starts CFX-Solver in batch mode (that is, without starting the CFX-Solver Manager interface).
`-bg-ccl <file>`		Reads Command Language from the named file, and uses it to provide defaults for the current run. If the file specifies a CFX-Solver input file for the run, the command language contained in that CFX-Solver input file will take precedence over that supplied. Also see the `-ccl` option.

Command-Line Options	Alternative form	Usage
`-ccl <file>`		Reads additional Command Language from the named file. Overrides any CFX Command Language specified in the CFX-Solver input file. If `<file>` is the single character '`-`', the Command Language is read from the standard input (usually the terminal). If any settings are made in the Command Language file which also occur on the command line to the left of the `-ccl` option, the settings in the file will take precedence, as stated above. This option may be repeated to include Command Language from more than one file.[a]
`-ccl2flow <executable>`		Starts <executable> instead of the standard ANSYS CFX `ccl2flow`.
`-cclsetup <executable>`		Starts <executable> instead of the standard ANSYS CFX `cclsetup`.
`-chdir <directory>`		Sets the working directory as specified.
`-check-only`		When running in batch mode, this will cause `cfx5solve` to verify its options, but exit before starting any processes, and is mostly for use by CFX-Solver Manager.
`-config <configuration name>`		Applies subsequent options to the specified configuration.
`-continue-from-file <file>`	`-cont-from-file <file>`	Uses initial values and continues the run from the specified CFX-Solver results file. The mesh from the CFX-Solver input file is used unless the `-use-mesh-from-iv` option is also specified. Only one -continue-from-file argument can be supplied. See Continuing the History (p. 82) in ANSYS CFX-Solver Modeling Guide for more details.
`-continue-from-configuration <configuration name>`	`-cont-from-config <configuration name>`	Uses initial values and continues the run from the most recent results for the named configuration. The mesh from the configuration (`.cfg`) file is used unless the `-use-mesh-from-iv` option is also specified. Only one -continue-from-configuration argument can be supplied. See Continuing the History (p. 82) in ANSYS CFX-Solver Modeling Guide for more details.
`-cplg-host <port@host>`		For an ANSYS Multi-field run, specifies CFX-Solver which **Port Number** and **Host Name** to use to communicate with the Mechanical application solver. For details, see MultiField Tab (p. 9).
`-definition <file>`	`-def <file>`	Uses `<file>` as the solver input file for a single configuration simulation. This may be a CFX-Solver input file or a CFX-Solver results file for a restart. The file specified is used in the same way as the input file on the **Define Run** dialog box. For details, see Define Run Command (p. 85). Also see the `-mdef` option.

Command-Line Options	Alternative form	Usage
-display <display>		(UNIX only) Uses the X11 server <display> instead of the X11 server defined by the DISPLAY environment variable.
-double		Starts the double-precision version of ANSYS CFX Partitioner, Interpolator and Solver. Also see the -single option.
-example <file>	-eg <file>	Starts the CFX-Solver using one of the example CFX-Solver input files provided with the product. The example StaticMixer is currently available.
-fullname <name>		Specifies the basename for the CFX-Solver output file, CFX-Solver results file, and the temporary directory based on <name> instead of the CFX-Solver input file name. No numerical suffix (such as _001) is added to the specified name.
-help	-h	Displays the help information for command-line options.
-initial <file>	-ini <file>	Uses the initial values in the CFX-Solver results file <file>. The mesh from this results file is used unless the -interpolate-iv option is also specified. This option has been deprecated and should be replaced by -initial-file or -continue-from-file as appropriate.
-initial-configuration <configuration name>	-ini-conf <configuration name>	Uses initial values from the most recent results for the named configuration as a basic initial guess for the run. The run history from this file is discarded. The mesh from this results file is used unless the -interpolate-iv option is also specified. See Continuing the History (p. 82) in ANSYS CFX-Solver Modeling Guide for more details.
-initial-file <file>	-ini-file <file>	Uses initial values from the specified CFX-Solver Results file as a basic initial guess for the run. The run history from this file is discarded. The mesh from the configuration (.cfg) file or the CFX-Solver input file is used unless the -use-mesh-from-iv option is also specified. See Continuing the History (p. 82) in ANSYS CFX-Solver Modeling Guide for more details.
-interactive	-int -manager	Starts CFX-Solver Manager in graphic user interface (GUI) mode. The CFX-Solver Manager interface allows an interactive control for starting a new run and/or manage or monitor an existing run.
-interpolate-iv	-interp-iv	Interpolates the solution from the initial values file, if one is supplied (using the -initial option), onto the mesh from the CFX-Solver input file, rather than using the mesh from the initial values file. This option has been deprecated and should be replaced by the -initial-file or -continue-from-file option, as appropriate.

Command-Line Options	Alternative form	Usage
-interp-double		When running with the solver-based interpolator (-interp-iv option), this option will select the double-precision version of the interpolator. It will not override the -interpolator option if both are used.
-interp-single		Uses the single precision ANSYS CFX Interpolator executable.
-interpolator <executable>		When running with the interpolator (-interp-iv option), this option will start <executable> instead of the default interpolator.
-job		Keeps the .job file after an ANSYS CFX Solver run. This file contains a brief summary of various solution values, and is most useful for regression purposes.
-job-part	-jobp	Keeps job file after an ANSYS CFX Partitioner run. This file contains a brief summary of various solution values, and is most useful for regression purposes.
-max-elapsed-time <elapsed time>	-maxet <elapsed time>	Sets the maximum elapsed time (wall clock time) that CFX-Solver will run. Elapsed time must be in quotes and have correct units in square brackets. For example: -maxet "10 [min]" or -maxet "5 [hr]".
-mdefinition <file>	-mdef <file>	Uses <file> as the solver input file. This may be a multi-configuration definition file or results file for a restart (that is, .mdef or .mres, respectively). The file specified is used in the same way as the input file on the **Define Run** dialog box. For details, see Define Run Command (p. 85).
-mfx-run-mode <mode>		Use this option to specify one of the following MFX run modes for an ANSYS Multi-field run: "Start ANSYS and CFX" "Start ANSYS only" "Start CFX only" "Process Input File only" For details, see MultiField Tab (p. 9).
-monitor <file>		When starting ANSYS CFX-Solver Manager, use this option to monitor the run represented by <file>, which may be a CFX-Solver results file or CFX-Solver output file.
-multiconfig		Treats the CFX-Solver input file as a multi-configuration input file.
-name <name>		Specifies the basename for exported files and the temporary directory based on the problem name <name> instead of the CFX-Solver input file name, unless other names are explicitly defined. This name cannot be set when using the CFX-Solver Manager to start the CFX-Solver.

Command-Line Options	Alternative form	Usage
-norun		Use this option to preprocess the CFX-Solver input file only, without running the solver executable. When used with a multi-configuration CFX-Solver input file, this option produces complete solver input files for the individual configuration (.cfg) files. When used with the "-config" option, only the specified configuration is preprocessed.
-par-dist <host-list>		Use this option to set the comma-separated <host-list> in the same form as is used in the Command Language definition. This option does not require the -partition option, as one partition is run on each host mentioned in the list. To run multiple partitions on the same host, it may be listed multiple times, or an asterisk may be used with the count, as in "wallaby*3,kangaroo*4" for a 7-partition run. Host details are taken from the hostinfo.ccl file, if they are there; otherwise, if possible, the required information will be automatically detected. <host> may be specified as [<user>@]<hostname>[:<CFX_ROOT>], if the user name or the ANSYS CFX installation root directory differs from the local host.
-par-host-list <host1> [,<host2> [,...]]		When running in parallel, uses the specified host list. See the -par-dist option for details of the host list. This option defaults to PVM Local Parallel on UNIX/Linux platforms and HP MPI Local Parallel on Windows.
-par-local		When running in parallel, uses only the local host. This will override the -par-dist or -par-host-list options.
-parallel	-par	Starts the solver in parallel mode. This option can be combined with -partition for a partitioning run. If the -part option is not specified, then the -parfile-read option must be used to specify a valid partitioning information file.
-parfile-read <parfile>		Specifies the name of an input partition file used to set up a partitioning or parallel run. **Note** Only *.par files that are generated in ANSYS CFX 12.0 (or later) are supported. For details, see CFX Partition File (p. 63).
-parfile-save		When used with a parallel run, saves the partitioning information to a file with the same basename as the results file, and the extension .par.
-parfile-write <parfile>		Specifies the name of a partition file to which to write the information from a partitioning run.

Command-Line Options	Alternative form	Usage
`-partition <number of partitions>`	`-part <number of partitions>`	Starts the solver in partitioning mode. This option should not be used if an existing partition file is also specified.
`-partitioner <executable>`		Starts `<executable>` instead of the standard partitioner.
`-part-coupled`		Activates coupled partitioning mode for multidomain problems. This is not activated by default.
`-part-independent`		Activates independent partitioning mode for multidomain problems. This is the default.
`-part-large`		Starts the large problem partitioner which can partition problems up to 2^31-1 elements. This partitioner uses 64-bit integer and logical variables so it will allocate more memory than the default partitioning executable.
`-part-mode <mode>`		Sets the partitioning mode to use when running the partitioner. Valid options are `metis-kway` (MeTiS k-way), `metis-rec` (MeTiS Recursive Bisection), `simple` (Simple Assignment), `drcb` (Directional Recursive Coordinate Bisection), `orcb` (Optimized Recursive Coordinate Bisection), `rcb` (Recursive Coordinate Bisection) Finer control over the partitioning method is available through the Command Language.
`-part-only <number of partitions>`		Starts the solver in partitioning mode only. This is normally equivalent to `-part`, but may be necessary if partitioning a results file from a previous run.
`-part-single`		Uses the single precision ANSYS CFX Partitioner. This is the default, but is provided for overriding any information which might be stored in the CFX Command Language dataset in a file from previous runs. Also see the `-double` option.
`-preferred-license <license name>`	`-P <license name>`	Sets the first license used by solver, given the availability of multiple usable licenses.
`-priority <level>`	`-pri <level>`	Enables the specification of a job priority to a solver run; the allowed values are `Idle` (`0`), `Low` (`1`), `Standard` (`2`), and `High` (`3`). The default value on UNIX platforms is `High`, which corresponds to nice increment of *0*. On Windows, the default is `Standard`, corresponding to a priority level of *Normal*.
`-save`		Use this option to avoid deleting any temporary files after the run. Normally the standard temporary files created by CFX-Solver are deleted automatically after each run.
`-serial`		Use this option to explicitly specify that a serial run is required. This is useful when restarting a run from a results file produced by a parallel run, where this option forces a serial run instead.

Command-Line Options	Alternative form	Usage
`-single`		Starts the single-precision version of the CFX-Solver, Partitioner, and Interpolator. This is the default, but is provided for overriding any information which might be stored in the CFX Command Language dataset in a file from a previous run. Also see the `-double` option.
`-size <factor>`	`-S <factor>` `-s <factor>`	Changes memory estimates used by the CFX-Solver by a factor of `<factor>`. Memory estimates are sometimes inaccurate and this option needs to be used to increase the memory allocated. For example, using `-size 1.05` increases memory allocated by 5%. This option provides the same functionality as set in **Solver Memory**. For details, see Solver Tab (p. 12). Further options for controlling the CFX-Solver memory allocation are available. Execute `cfx5solve -help` for full details.

`-size-cat <size>`
`-size-nr <size>`
`-size-ni <size>`
`-size-nd <size>`
`-size-nc <size>`
`-size-nl <size>`
	`-scat <size>`
	`-nr <size>`
	`-ni <size>`
	`-nd <size>`
	`-nc <size>`
	`-nl <size>`
		These flags are for advanced users to change the memory allocation parameters for the solver. Usually, you should use the `-size` option instead. `<size>` is the desired memory allocation in words, and may have K or M appended for *kilo* or *mega*. If the suffix is 'x', then the number is treated as a multiplier.

`-size-cclsetup <factor>`	`-sizeccl <factor>`	Changes the memory estimates used by the ANSYS CFX `cclsetup` executable by a factor of <factor>.

`-size-cclsetup-cat <size>`
`-size-cclsetup-nr <size>`
`-size-cclsetup-ni <size>`
`-size-cclsetup-nd <size>`
`-size-cclsetup-nc <size>`
`-size-cclsetup-nl <size>`
	`-scatccl <size>`
	`-nrccl <size>`
	`-niccl <size>`
	`-ndccl <size>`
	`-ncccl <size>`
	`-nlccl <size>`
		These options are the same as the `-size-*` options above, but provide sizes needed for the ANSYS CFX CCL Setup executable.

`-size-interp <factor>`	`-sizeint <factor>`	Changes the memory estimates used by the solver-based interpolator by a factor of `<factor>`. Also see the `-size` option.

`-size-interp-cat <size>`
`-size-interp-nr <size>`
`-size-interp-ni <size>`
`-size-interp-nd <size>`
	`-scatint <size>`
	`-nrint <size>`
	`-niint <size>`
	`-ndint <size>`
	`-ncint <size>`
	`-nlint <size>`
		These options are the same as the `-size-*` options above, but provide sizes needed for the ANSYS CFX Interpolator.

Command-Line Options	Alternative form	Usage
-size-interp-nc <size> -size-interp-nl <size>		
-size-mms <factor>	-smms <factor>	Changes the initial MMS catalogue size estimate used by the CFX-Solver by a factor of <factor>. This option has been deprecated and should be replaced by -size-cat.
-size-part-mms <factor>	-smmspar <factor>	Changes the initial MMS catalogue size estimate used by the partitioner by a factor of <factor>. This option has been deprecated and should be replaced by -size-part-cat.
-size-part <factor> -size-par <factor>	-sizepart <factor> -sizepar <factor>	Changes the memory estimates used by the ANSYS CFX Partitioner by a factor of <factor>. Also see the -size option. Further options for controlling the partitioner memory allocation are available. Execute cfx5solve -help for full details.
-size-part-cat <size> -size-part-nr <size> -size-part-ni <size> -size-part-nd <size> -size-part-nc <size> -size-part-nl <size>	-scatpar <size> -nrpar <size> -nipar <size> -ndpar <size> -ncpar <size> -nlpar <size>	These options are the same as the -size-* options, but provide sizes needed for partitioner rather than solver.
-solver [<os>=]<executable>, <os>=<executable>[...]]	-exec [<os>=]<executable> [,<os>=<executable>[, ...]]	Starts <executable> instead of the standard ANSYS CFX solver on <os>, where <os> is the short architecture string for the desired operating system. If <os> is omitted, then the current operating system is assumed. For example: the command-line option -solver "linux-amd64/mysolver.exe,linux=linux/mysolver.exe" uses the executable "linux-amd64/mysolver.exe" for the current operating system and the executable "linux/mysolver.exe" for the "linux" operating system. Full paths or paths relative to the working directory may be used when specifying solver executables. In this example, it is worth noting that the current operating system is presumed to be "linux-amd64", and that the "linux-amd64/mysolver.exe" and "linux/mysolver.exe" will be used for all solvers running on "linux-amd64" and "linux" operating systems, respectively. The string value for <os> can be determined by running the following command:

Command-Line Options	Alternative form	Usage
		• On Unix-like systems, execute `<CFXROOT>/bin/cfx5info -os`. • On a Windows system, execute `<CFXROOT>\bin\cfx5info -os`. where `<CFXROOT>` is the path to your installation of ANSYS CFX.
`-solver-single`		Uses the single precision ANSYS CFX Solver.
`-solver-double`		Uses the double precision ANSYS CFX Solver.
`-start-method <name>`		Uses the named start method to start the solver. This option allows you to use different parallel methods, as listed in the CFX-Solver Manager GUI, instead of the defaults. For parallel runs, you also need to provide the `-part` or `-par-dist` options.
`-use-mesh-from-iv`		Uses the mesh from the source initial values (that is, from a file or configuration) rather than from the solver input file. This is only valid if a single initial-values source is specified.
`-verbose`	`-v`	Specifying this option may result in additional output being sent to the standard output file (normally the screen).

[a]Locators for applying physics cannot be modified using the `-ccl` option; they can, however, be changed in CFX-Pre.

Command-Line Samples

Here are some examples to help clarify the use of the command line:

Start CFX-Solver

To start CFX-Solver running from the CFX-Solver input file `model.def`, enter the command:

```
cfx5solve -def model.def
```

If the input file is for a multiple configuration (`.mdef` file), enter the command:

```
cfx5solve -mdef model.mdef
```

Start CFX-Solver Manager

To start CFX-Solver Manager, passing it the name of the CFX-Solver input file, enter the command:

```
cfx5solve -interactive -def model.def
```

Produce a Partition File

To produce a partition file with the `MeTiS` partitioning method and seven partitions, but *not* run CFX-Solver to solve for the solution, enter the command:

```
cfx5solve -def model.def -partition 7
```

This command will produce a file named `model_001.par` in the local run directory.

Note

If the file `model.par` exists in the working directory, then the partition type (`MeTiS`, `RecCoordBis` or `SpecDir`) is read from this file, even if you have not specified the file `model.par`. Because this could potentially be confusing, you are advised to use the CFX-Solver Manager to set up a partitioning run, unless you are certain that either there is no file `model.par` or that the partitioning method specified in the `model.par` file is what you require.

Start CFX-Solver in Local Parallel

To run CFX-Solver in parallel, starting from the CFX-Solver input file `model.def` and running only on the local machine with two partitions, enter the command:

```
cfx5solve -def model.def -par-local -partition 2
```

If you have already created a file `model.par` (for instance, by using the command `cfx5solve -def model.def -partition 7`), then you can run the parallel CFX-Solver by entering the command:

```
cfx5solve -def model.def -par-local -parfile-read model.par
```

To run the CFX-Solver in parallel for the configuration named `<config>` and in serial for other configurations in a multi-configuration simulation, enter the command:

```
cfx5solve -mdef model.mdef -config "<config>" -par-local -partition 2
```

Start CFX-Solver in Distributed Parallel

For this example to work, the hosts `hosta`, `hostb` and `hostc` must be defined in the central `hostinfo.ccl` file.

To run CFX-Solver in distributed parallel, starting from the CFX-Solver input file `model.def`, and using 1 partition on `hosta`, 2 partition on `hostb` and 4 partitions on `hostc`, for a total of 7 partitions, enter the command:

```
cfx5solve -def model.def -par-dist 'hosta,hostb*2,hostc*4'
```

Start CFX-Solver in Parallel

To start the CFX-Solver in parallel with four partitions on two hosts, enter the command:

```
cfx5solve -def file.def -par-dist 'hosta*2,hostb*2'
```

If you have already created a partitioning file, say model.par (for instance, by using the command `cfx5solve -def model.def -partition 7`), then you can execute the distributed parallel run as follows:

```
cfx5solve -def model.def –parfile-read model.par -par-dist 'hosta,hostb*2,hostc*4'
```

The number of partitions specified using the `–par-dist` command-line flag must be the same as that in the partitioning file, `model.par`, or the run will fail.

Start CFX-Solver in Parallel Using MPICH2

To start the CFX-Solver in parallel, with two partitions, using MPICH2 Local Parallel on a Windows machine (give this command as a single line), enter the command:

```
cfx5solve -def file.def -start-method "MPICH2 Local Parallel for Windows" -part 2
```

Start an ANSYS Multi-field Run (FSI)

To start an ANSYS Multi-field Run, launching both the Mechanical application and CFX Solvers, starting from the CFX-Solver input file `model.def` and using the Mechanical application input file `model.inp`:


```
cfx5solve -def model.def -ansys-input model.inp
```

To start an ANSYS Multi-field run, launching the Mechanical application solver only, disabling the processing of the Mechanical application input file and using the Mechanical application input file `model.inp`:

```
cfx5solve -def model.def -ansys-input model.inp -mfx-run-mode "Start ANSYS only" -ansys-
```

To start an ANSYS Multi-field run, launching CFX-Solver only and telling it to communicate with the already running Mechanical application solver on a particular port number and host, enter the command:

```
cfx5solve -def model.def -mfx-run-mode "Start CFX only" -cplg-host 49800@machine1 -cplg-
```

Preprocess Incomplete Configuration Files

Configuration definition `(.cfg)` files that are created in conjunction with a multi-configuration simulation file `(.mdef)` are incomplete; they do not contain global information like equation and material definitions.

To preprocess the configuration files so that they are complete and can be run independent of the multi-configuration simulation, enter the command:

```
cfx5solve -mdef model.mdef -norun
```

To preprocess the configuration definition file corresponding to the configuration name `<config>`, enter the command:

```
cfx5solve -mdef model.mdef -config "<config>" -norun
```

Stopping the CFX-Solver from the Command Line

After CFX-Solver is running, stop it by using `cfx5stop` from the command line.

Suppose a run is called `mixer_001` in the current directory. There will be a temporary directory called `mixer_001.dir` in the current directory while that run is actually running. To stop the run, enter the following command line:

```
cfx5stop -directory mixer_001.dir
```

Chapter 12. CFX-Solver Start Methods

CFX-Solver Start Methods define allowable parameters and command-line arguments used by CFX-Solver Manager and CFX-Solver Script to launch the CFX-Solver executable and perform a run. The definition of the solver start methods are modifiable by expert users to customize solver start-up for specific parallel or batch queuing environments.

The standard start methods for a CFX installation are contained in the `<CFXROOT>/etc/start-methods.ccl` file. These can be over-ridden by placing a custom version of this file in the same location as the site or user CFX configuration files.

Direct Start Methods

Solver start methods that directly run the CFX-Solver executable are known as *Direct* start methods, and are commonly used for defining solver execution in serial or parallel on local networks. Definition of solver start methods for specific parallel environments can be made following the various available PVM (UNIX only) and MPI methods in the file.

Indirect Start Methods

Solver start methods can also be used to launch a user-defined script or program that can perform system interaction before re-executing a `cfx5solve` command under a different environment or system. These are known as indirect start methods, and are commonly used for executing CFX-Solver runs on remote or batch queuing systems. Use of Indirect start methods requires knowledge of a system scripting language (such as bash or Perl) to customize the solve start-up process for your specific environment.

Example

An example of the use of Indirect Solver Start methods to run a `cfx5solve` command within a PBS-based batch queuing system is provided in the CFX distribution. For details, please see the *Submit to PBS Queue* start method in `start-methods.ccl`, and the `qcfx5solve.pl` script in the `<CFXROOT>/extras/` directory.

Chapter 13. CPU and Memory Requirements

This chapter provides information on typical increases in CPU (central processing unit) time and memory requirements incurred by some simulations and physical models:

Tetrahedral Mesh

The ratio of elements to nodes is approximately 5:1 for a tetrahedral mesh. For example, if 5 million tetrahedral elements are in a mesh, then there are approximately 1 million nodes. This is in contrast to a hexahedral mesh where the ratio of elements to nodes approaches 1:1 as the grid becomes large.

Memory required for a tetrahedral mesh is about 0.4 times the memory required for a hex mesh of the same number of elements. Alternatively a tetrahedral mesh has about 2 times the required memory of a hexahedral mesh with the same number of nodes.

Special Partitioner, Solver and Interpolator Executables

On all systems there are default Partitioner, Solver and Interpolator executables. On some systems, there are additional executables. There are currently two classes of special executables:

Double-Precision Executables

Double-precision executables store basic floating point numbers as 64 bit words. These executables are available to permit more accurate numerical mathematical operations. Double precision accuracy might be needed if the computational domain involves a huge variation in grid dimension, aspect ratio, pressure range, etc.

When double precision is used, the computer memory used for a given problem and grid size is double that of the default (single precision) executable. Stated another way: the maximum problem size to run on a given computer for the double precision executable is half that of the default single precision executable.

Large Problem Partitioner Executables

This special executable is only available on 64-bit platforms. The standard partitioner is currently limited to allocate 2^31-1 words of 4 byte integer stack space which limits the maximum problem size for partitioning to approximately 80 million elements (structured) and 200 million elements (unstructured).

As a workaround, larger problems can be partitioned with the large problem partitioner. This executable internally uses 8 byte integer data to perform the partitioning process. In theory, a maximum problem size of 2 billion elements can be partitioned with this executable. However, practical considerations, such as available computer resources, will still limit the maximum size.

Turbulence

The following topics will be discussed:

Zero Equation Model

The use of this model incurs a small increase in CPU time and memory requirements compared to laminar flow.

Two-Equation Models

Two additional scalar equations are solved when using two-equation turbulence models. The SST model has a slight additional cost over other two equation models since a wall scale equation is also solved.

Consider the case of single phase, single component laminar flow in which the U-Mom, V-Mom, W-Mom, and P-Mass equations are solved. Expect a CPU cost increase on the order of 50% by the addition of a two-equation turbulence model. Memory requirement increases are small.

Reynolds Stress Model

This model adds six scalar equations for each of the Reynolds Stresses as well as the Eddy Dissipation equation. It is approximately 2.5 times more expensive than the two equation turbulence models.

Consider the case of single phase, single component laminar flow in which the U-Mom, V-Mom, W-Mom, and P-Mass equations are solved. Expect a CPU cost increase on the order of 120% by the addition of a Reynolds Stress turbulence model. Memory requirement increases are small.

Energy Models

Both the thermal and total energy models require the solution of an additional scalar equation. The solution of the energy equation typically requires 1/3 of the CPU required for the U-Mom, V-Mom, W-Mom, and P-Mass equations. Memory requirement increases are small.

CHT Regions

Only the energy equation is solved in CHT regions, so compared to the same number of nodes in a fluid region, the CPU costs are much less (U-Mom, V-Mom, W-Mom and P-Mass are not solved).

Multicomponent Flows

Each additional component adds an extra scalar equation. Therefore, as the number of components increase, CPU time required to solve the Mass Fraction equation increases linearly. Expect each component to add approximately 25% to the CPU required for the U-Mom, V-Mom, W-Mom and P-Mass equations.

Multiphase Flows

The following topics will be discussed:

- N-Phase Flow (p. 125)

Homogeneous Model

For two-phase flow using the homogeneous model, expect memory requirements to increase by a factor of 1.5 and CPU time to increase by a factor of 1.7 over the same single-phase simulation. Enabling free surface does not significantly alter CPU or memory requirements.

Inhomogeneous Model

For two-phase flow using the particle or mixture models, expect memory requirements to increase by a factor of 2.15 and CPU time to increase by a factor of 2.25. Enabling free surface does not significantly alter CPU or memory requirements.

N-Phase Flow

As the number of fluids increase, expect memory and CPU requirements to increase approximately linearly for small N. Tetrahedral meshes have more of a linear increase than hexahedral meshes. The table below gives approximate memory increase factors for up to 5 phases when compared to the same single-phase simulation on a hexahedral mesh.

# of Phases	Memory Increases	
	(Hex Mesh)	(Tet Mesh)
1	1	1.80
2	2.15	3.40
3	3.50	5.70
4	5.15	8.05
5	7.00	10.60

Expect the CPU factor to be slightly less than the corresponding memory factors, but the trend is the same.

Additional Variables, Wall Distance Variables, and Boundary Distance Variables

A single scalar equation is added for each Additional Variable. A single scalar equation is also added whenever the Wall Distance variables or Boundary Distance variables are referenced in a CFX Expression Language (CEL) expression, or when these quantities are required for a built-in model (for example, two equation-turbulence or boundary-distance-based mesh-stiffness models). Note that Wall Distance values and Boundary Distance values are derived from the solutions of the Wall Scale equations and Boundary Scale equations, respectively. Detailed information on the wall scale equation is available at Wall and Boundary Distance Formulation (p. 50) in ANSYS CFX-Solver Theory Guide.

Expect a CPU cost increase of approximately 20% for each Additional Variable over the solution of the U-Mom, V-Mom, W-Mom, and P-Mass equations for a single-phase single-component case. Increases in memory requirements are small.

Combustion Modeling

Modeling combustion incurs a slight cost compared to multicomponent flow with the same number of components. For multi-step reactions each component is solved using the coupled solve. This incurs additional CPU time which does not increase linearly with the number of components.

Radiation Modeling

This adds a single scalar equation. Cost increases are similar to those of the energy equation. For details, see Energy Models (p. 124).

GGI Interfaces

An intersection algorithm which is performed at the start of a simulation to connect each side of a GGI connection incurs a one-time cost.

Each GGI connection means approximately 5% more CPU time and memory is required. This number can vary greatly, as it is a function of the number of nodes involved in a GGI connection, compared to the number of nodes that are not involved in the GGI connection. There is also a dependence on the geometric complexity of the GGI connection.

Transient Runs

Each coefficient loop requires approximately the same CPU time as the equivalent steady-state iteration.

Mesh Deformation

Mesh deformation using either of the `Regions of Motion Specified` or `Junction Box Routine` options introduces several CPU intensive operations during each outer iteration or time step. When deformation is performed using `Regions of Motion Specified`, a Mesh Displacement equation is assembled and solved at the start of each outer iteration or time step for Steady State and Transient simulations, respectively, and the mesh coordinates are updated. When deformation is performed using a `Junction Box Routine`, you define how mesh coordinates are updated.

After updating the mesh coordinates, other mesh related quantities (such as volumes, areas, mesh quality measures, mesh velocities, etc.) are updated, and GGIs are re-intersected before advancing to solve other equations or physical models for the current outer iteration or time step. Mesh volume flows that are later used to augment mass flow rates applied in advective transport are precalculated and stored during these updates.

Depending on the complexity of the deformation and physical model (for example, the use of GGIs), adding the mesh deformation to a simulation will increase CPU usage by approximately 10% to 50% per outer iteration or time step.

Adding the mesh deformation will increase memory requirements due to the storage of: the noted mesh volume flows (one per control volume integration point), and multiple sets of mesh coordinates for transient simulations (one triplet per mesh vertex per time step that must be kept for the selected transient discretization).

Bi-Directional (Two-Way) Couplings with ANSYS Multi-field

Enabling an external solver coupling with ANSYS Multi-field introduces a coupling (or stagger) iteration layer in addition to the time step and coefficient loop iteration structure used for simulations involving CFX only. As outlined in Bidirectional (Two-Way) FSI (p. 76), time steps are executed using a sequence of coupling iterations, which involve one or more coefficient loop iterations within either the CFX or ANSYS field solver.

Additional memory is not required when external solver couplings are used. However, additional CPU time is required because of the CFX solver coefficient loops performed per coupling iteration. In general, the CFX solver CPU usage will increase by a factor that is slightly smaller than the number of coupling iterations required per time step. A summary of expected CPU time increases is tabulated below, according to the degree of coupling between the fluid and solid physical problems.

Degree of Coupling	**CPU Increase**
Weak	2× to 5×

Degree of Coupling	**CPU Increase**
Typical	5× to 10×
Strong	> 10×

Chapter 14. The cfx5control Application

The cfx5control application can be used to dynamically control the CFX-Solver. The features available include:

- Stopping the solver running in the given directory at the end of the current timestep:

  ```
  <CFXROOT>/bin/cfx5control <directory> -stop
  ```

- Instructing the solver running in the named directory to write a backup results file.

  ```
  <CFXROOT>/bin/cfx5control <directory> -backup
  ```

- Editing the Command Language during a run:

  ```
  <CFXROOT>/bin/cfx5control <directory> -edit-commands [-no-backup]
  ```

- Reading Command Language from a file and implementing it on the fly.

  ```
  <CFXROOT>/bin/cfx5control <directory> -inject-commands <file> [-no-backup]
  ```

- Adjusting the priority of a CFX-Solver run by resetting the run priority on Windows or altering the *nice* increment on non-Windows platforms. This applies to all solver processes in a parallel run.

  ```
  <CFXROOT>/bin/cfx5control <directory> -reset-priority <level>
  ```

  where `<level>` is one of `Idle`, `Low`, `Standard` or `High`, as given in the following table:

CFX Run Priority Level		UNIX nice inc.	Windows Priority
Idle	0	19	Low
Low	1	7	BelowNormal
Standard	2	0	Normal
High	3	0	AboveNormal

  If the current priority level is the same as `<level>` then there is no change. Administrative (or root) privileges are usually required to increase the priority from a lower level to a higher level. When the change of priority is attempted, then the CFX-Solver will write a diagnostic message into the CFX-Solver Output file of the form:

  ```
  +--------------------------------------------------------------------+
  |            ****** Updating Runtime Priority ******                 |
  |                                                                    |
  |      <outcome of the attempt to change the run priority>           |
  |                                                                    |
  +--------------------------------------------------------------------+
  ```

- Displaying help for this command:

  ```
  <CFXROOT>/bin/cfx5control -help
  ```

In these examples:

- `<CFXROOT>` is the path to your installation of CFX
- `<directory>` specifies a directory in which the ANSYS CFX solver is currently running, such as StaticMixer_004.dir.
- `-no-backup` prevents the solver from writing a backup file before reading the new Command Language file.

Chapter 15. File Export Utility

The ANSYS CFX results file generated by a CFX-Solver run contains the details of the mesh used to perform the calculation as well as details of the results variables that have been calculated. For details on which variables an ANSYS CFX results file contains, see List of Field Variables (p. 162).

To postprocess an ANSYS CFX results file using software other than CFD-Post, the mesh and variables can be extracted from the ANSYS CFX results file into a format compliant with the 3rd party software. ANSYS CFX provides predefined translations to a number of different postprocessors and analysis packages, and CFX-Solver Manager provides an interface to enable easy translation to these formats. A command line utility, cfx5export, can be used to perform the same operations in batch mode.

The standard file formats that can be generated from ANSYS CFX are suitable for direct input into the following software systems:

- ANSYS Multi-field
- All systems that support CGNS files
- MSC.Patran, from the MacNeal-Schwendler Corporation
- FIELDVIEW, from Intelligent Light
- EnSight, from Computational Engineering International, Inc.

It is also possible to write to other formats by creating a customized export program. See Creating a Customized Export Program (p. 37).

When using CFX-Solver Manager, it is possible to export the results of a CFX-Solver run by selecting either **Tools > Export to ANSYS Multifield** (see Export to ANSYS Multi-field solver Dialog (p. 131)) or **Tools > Export** (see Export of Results to Other Formats (p. 132)).

This chapter describes:

Alternatively, it is possible to export the results directly from the command line. For details, see Running cfx5export from the Command Line (p. 143).

Export to ANSYS Multi-field solver Dialog

The **Export to ANSYS Multifield solver** dialog is used to produce ANSYS `.cdb` files for use in Fluid Structure Interaction cases. To display the dialog, select **Tools** > **Export to ANSYS Multifield**.

The sections that follow describe the **Export Options** fields on the dialog.

ANSYS Element Type

Selections correspond to the type of ANSYS element to export. The choices are 3D Thermal (element type 70), 2D Thermal (element type 152), and 2D Stress (element type 154). Refer to the *Theory Reference for Mechanical* for a description of these element types.

Output Modifiers

The following scaling and offset factors can be used to change the units of the solution. The factors can be applied when you need to convert the units that are written to the CFX file into those that you want to use in ANSYS.

The available modifiers are:

Offset Flow

Default Value: 0

Requires a real number corresponding to the offset value applied to solution flows.

Offset Values

Default Value: 0

Requires a real number corresponding to the offset value applied to solution values.

Scale Flow

Default Value: 1

Requires a real number corresponding to the scaling factor applied to solution flows.

Scale Values

Default Value: 1

Requires a real number corresponding to the scaling factor applied to solution values.

Export of Results to Other Formats

To export CFX-Solver result to formats such as CGNS, EnSight, FIELDVIEW, MSC.Patran, or another customized format, select **Tools > Export** from the menu bar of CFX-Solver Manager. The following section details the common options that can be used when exporting results to the standard formats supported by ANSYS CFX.

Generic Export Options

To configure data for export, the **Export** dialog box must be displayed: select **Tools** > **Export**.

There are numerous export options when writing to the supported 3rd-party formats:

Results File

The name of the CFX-Solver results file to be exported.

Domain Selection:Name

The domain or domains in the CFX-Solver results file that are to be exported.

Timestep Selection:Time

For transient CFX-Solver results files, the timestep to be exported.

Output Format

Default Value: CGNS

The output format selects which standard file format the output file will be written in. If a nonstandard format is required, select **Custom User Export**.

The sections that follow describe the available output formats.

> **Note**
>
> For simulations with multiple configurations, `*.mres` files cannot be exported by the CFX-Solver Manager.

CGNS

The CFD General Notation System (CGNS) is designed to facilitate the exchange of data between sites and applications, and to help stabilize the archiving of data. The data is stored in a compact, binary format.

> **Note**
>
> To configure this option, select **Tools** > **Export**. The dialog box uses numerous common export options. For details, see Generic Export Options (p. 132).

This section contains:

Overview

CGNS consists of a collection of conventions for the storage and retrieval of CFD data. The system consists of two parts:

- A standard format in which the data is recorded.
- Software that reads, writes, and modifies data in that format.

Available Options

The following options are available when writing a file in CGNS format:

- All options specified in Generic Export Options (p. 132)
- **Geometry Output Only**
  - *Default Value:* Unselected.
  - If selected, only mesh information and boundary condition information is written to the destination file—result variables are not written.
- **Output boundaries as**
  - *Default Value:* Nodes
  - If **Nodes** are selected, all boundary conditions are written as collections of nodes; if **Faces**, then boundary conditions are written as groups of 2D elements (faces). CFD-Post cannot read boundaries that are output as **Faces**.

- When **Write Transient Data to One File** is selected, all transient data (grid and results) are written to one CGNS file using `BaseIterativeData_t` and `ZoneIterativeData_t` nodes within the file. When this option is not selected, transient results are exported to a separate file for each timestep.
- When **Use CGNS Variable Names** is selected ANSYS CFX variable names are mapped to CGNS variable names. For example `Total Pressure` becomes `PressureStagnation`.

Exported Files

The exported file set consists of either a single file for non-transient results, or multiple files for transient results. Each contains a complete mesh and flow solution. By default all files are generated with a `.cgns` extension. Import into a program that reads CGNS files should be done according to the importing program's instructions.

ANSYS CFX-Export is capable of writing CGNS Version 2.4 files. These CGNS files can be read by third parties if they support the features CFX writes and are using CGNS Version 2.0 and above.

> **Note**
>
> A CGNS-valid file can be written in ADF or HDF5 format. CFX currently supports only ADF base files. When HDF5 becomes standard practice, it is expected that both versions will be supported and identified seamlessly.

Contents of CGNS Files Written by ANSYS CFX

The file produced contains grid and solution data. It does not contain problem setup (physics) information.

The amount of solution data and the type of grid written to the CGNS file is user controllable either on the command line or via the GUI. What is seen in the CGNS file reflects what the user requests. There are files that when written using some options may not be able to be reread into CFX-Pre. Caution must therefore be used if the original CFX solution files are deleted, as it may not be possible to recover all information.

Names of variables, zones, and boundary conditions are always CGNS-compliant. The name seen within the CFX application may have to be changed to achieve this.

The remainder of this sections describes the data records that are written when creating a CGNS file from ANSYS CFX.

Base (Base_t)

A single base is written to each CGNS file.

- Its name is not significant.
- It is always written with a cell_dimension of 3 (that is, 3D).

If a transient file is being written, a simulation type (`SimulationType_t`) of `TimeAccurate` is specified below the base node.

A state (`ReferenceState_t`) is also written below the base node with a description of where the file was generated from and what it represents.

Zones (Zone_t)

A single zone is written under each `Base_t` node for each domain requested by the user and is always unstructured in nature.

Coordinates of node data are always written in double precision. Due to the nature of the grid being unstructured, there is no implicit ordering in how the grid is written.

Elements (Elements_t)

Element sections are written on a one per domain/subdomain basis as well as a one per boundary condition basis. Due to the nature of CFX data, a single element cannot appear in more than one element section. Element numbering is unique and consecutive.

You can control whether volume mesh (with surface mesh) or a surface mesh is written to the file.

Boundary Conditions (BC_t)

Boundary Conditions are written. The location of each boundary condition is specified as set of 2D elements (faces) or a set of nodes. The former is generally preferred as the latter can have some restrictions for the program that reads the file.

No properties of the boundary condition are written other than its type.

Solution Data (FlowSolution_t)

Solution data is written where requested. Names are mapped to be CGNS compliant. No discrete data is currently written. Where `Wall Only` data is present, “missing” solution data is written as 0.0.

Transient Data

Transient data is written to separate CGNS files by writing the Grid and Solution data for each CFX transient file that is available to the cfx5export process.

ANSYS CFX Connectivity using CGNS for Aerodynamic Noise Analysis

Further information on exporting files is contained in Aerodynamic Noise Analysis (p. 311) in the ANSYS CFX-Solver Modeling Guide.

Reading Exported Files into a Program Supporting CGNS

There is a wide range of products that can import CGNS files. Consult user documentation for the product being used for further information.

> **Note**
>
> An issue was detected while reading a CGNS file in TecPlot 10 and earlier that prevented the files written by ANSYS CFX being read by TecPlot. If a problem is encountered, try setting the environment variable CFX5_EXPORT_CGNS_TECPLOT to a value of 1, restart ANSYS CFX, and export the CGNS file again. If the problem persists, contact either ANSYS support or TecPlot support.

MSC.Patran

MSC.Patran is a general-purpose CAE simulation tool.

> **Note**
>
> To configure this option, select **Tools** > **Export**. The dialog box uses numerous common export options. For details, see Generic Export Options (p. 132).

This section contains:

Available Options

The following options are available when writing a file in MASC.Patran format:

- All options specified in Generic Export Options (p. 132)
- **Geometry Output Only (Neutral File)**
  - Default Value: Unselected.

    If selected, the mesh from the ANSYS CFX results file is written to a Neutral file; solution variables are not written to the Neutral file. For more details about MSC.Patran Neutral files, refer to your documentation from MSC.Patran.

Exported Files

The file set for this export option consists of three files:

File Name	File Type
<basename>.out	PATRAN Neutral File
<basename>.nod	PATRAN 2.5 Nodal Results File
<basename>.results_tmpl	PATRAN Results Template File

ANSYS CFX writes files in ASCII format using a subset of the record types to be found in the full definition of the PATRAN file formats. The full definition of the PATRAN file formats can be found in the MSC.Patran documentation.

Faces associated with the CFX boundaries and elements associated with subdomains are transferred into PATRAN named components. The boundary/subdomain components are named using their CFX name.

Exported nodes are associated only with the PATRAN default_group, unless `-nodes` is used when cfx5export is used from the command line.

Reading Exported Files into MSC Patran

The following procedure should be used for successfully import of results into MSC.Patran Version 2001r2. However, this may need to be adapted depending on what MSC.Patran is used for. For more details about reading MSC Patran Neutral files, see the MSC.Patran documentation.

> **Note**
>
> An MSC.Patran warning may appear when importing the Neutral file that reads *No element type could be found for element property set <P_SET.1>*. No action needs to be taken.

1. Prepare the required PATRAN files.

   Run cfx5export on the CFX results file, either using CFX-Solver Manager or directly from the command line.

2. Start PATRAN and create a new database.

   Use **File** > **New** to create a new database. Click on **Enable NFS access** (on UNIX systems only). Enter a new database name and click **OK**.

   In **New Model Preferences**, choose the appropriate Analysis Code option for the analysis.

3. Import CFX-Mesh and results.

   Use **File** > **Import** to access **Import**. Set **Object/Source** to `Model/Neutral`. Enter the name of the neutral file produced by ANSYS CFX and click **Apply**. Acknowledge the first pop-up and answer **Yes** to the second to continue.

   Use **Import** again with **Object/Source** set to `Results/PATRAN2.nod`, to read the nodal results file produced by cfx5export. **Template for PATRAN 2.5 Import Results** appears. Enter the name of the template file produced by cfx5export, and click **OK**. Enter the nodal results file name in Import and click **Apply**.

4. Continue to use MSC.Patran as required.

Exporting Boundary Conditions to MSC.Patran

CFX can be used to provide data to be used as boundary conditions for other types of analysis in MSC.Patran.

MSC.Patran enables models to be prepared for several different kinds of analysis. It also has facilities for using imported data to define data fields suitable for interpolating loads and boundary conditions onto the geometry or the mesh of a new model.

A description of the relevant PATRAN facilities, in particular the **Fields** function and its applications, can be found in the MSC.Patran documentation.

Example Procedure

Here is an outline of some guidelines for one possible procedure for incorporating CFX results into a PATRAN model. CFX temperature predictions will be used to define a temperature distribution on a geometry surface of a new model. Details about any of the options can be found in the MSC.Patran documentation.

Points to note in this example are:

- Files generated by ANSYS CFX should be read into a new database first before any PATRAN model grid. This ensures that the node numbers of the Neutral file correspond to the nodal result file. The nodes and elements in the new model mesh will be numbered or renumbered to follow on from those in the CFX data.
- The CFX model should not be included in the new model analysis, but the CFX data must not be deleted until the boundary values have been interpolated onto the new grid.
- The new model mesh can either be created or imported as part of step Step 6 (p. 137) or in a later step, without affecting the procedure, because in this example the boundary condition is defined on geometry surfaces.

The example assumes exported results include the Temperature variable.

1. Prepare PATRAN Neutral and Nodal results files containing boundary data only.

   Export the ANSYS CFX results file to MSC.Patran format, either using CFX-Solver Manager or directly from the command line. Toggle **Boundary Data Output Only** on if using CFX-Solver Manager, or use the `-boundary` option if using cfx5export from the command line.

2. Start PATRAN and create a new database.

   Use **File** > **New** to create a new database. Click on **Enable NFS access** (on UNIX systems only). Enter a new database name and click **OK**.

   On **New Model Preferences**, choose the appropriate Analysis Code option, for example, `PATRAN 2 NF`.

3. Import CFX mesh and results.

   Use **File** > **Import**. Set **Object/Source** to `Model/Neutral`. Enter the name of the Neutral file produced by ANSYS CFX and click **Apply**. Acknowledge the pop-up to continue.

   Use Import again with **Object/Format** set to `Results/PATRAN2.nod`, to read the nodal results file produced by ANSYS CFX. **Template for PATRAN 2.5 Import Results** appears. Enter the name of the template file produced by ANSYS CFX, and click **OK**. Enter the nodal results file name and click **Apply**.

4. Display temperature results as a fringe plot.

   Select **Group/Post** to post just the group containing the boundary condition nodes to use. Select **Results**. On **Results**, select the temperature results, and click **Apply**.

5. Create a continuous FEM field from the displayed variable.

   Select the **Fields** option. Set **Action/Object/Method** to `Create/Spatial/FEM`. Enter a new name under **Field Name**. Click the **Continuous** option. Select the relevant group under **Mesh/Results Group Filter**. Click **Apply**.

6. Add the geometry for the new model.

   Post the default_group and make it current using **Group/Post**. Use the toolbar icons to ensure the display will be in wireframe mode rather than in hidden line mode. Then either create a geometry for the new PATRAN model using the **Geometry** option to open the **Geometry** form or import the geometry from a previously prepared database using **File** > **Import** with **Object/Source** set to `Model/ MSC/PATRAN DB`.

7. Define a temperature boundary condition on new geometry surfaces.

   Select the **Load/BCs** option. Set **Action/Object/Type** to `Create/Temperature/Nodal`. Enter a name under **New Set Name**. Click **Input Data**. Select the field in the **Spatial Fields** box and click **OK**. Under **Load/Boundary Conditions**, click **Select Application Region** to display **Select Application Region**. Click the **Geometry** option under **Geometry Filter**. In **Select Geometric Entities**, pick the surfaces to apply the boundary condition to. Click **Add** and then **OK**. In **Load/Boundary Conditions**, click **Apply**. A temperature distribution should now be visible on the relevant geometry surfaces, in the form of values shown at the intersections of the surface display lines.

8. Complete the PATRAN model.

FIELDVIEW

FIELDVIEW is a standalone CFD post processor supplied by Intelligent Light.

> **Note**
>
> To configure this option, select **Tools** > **Export**. The dialog box uses numerous common export options. For details, see Generic Export Options (p. 132).

This section contains:

Available Options

The following options are available when writing a file for use in FIELDVIEW Unstructured format:

- All Options specified in Generic Export Options (p. 132).
- **FV-UNS File Options**: The options in this section detail the formatting of the separate grid and results files or combined grid and results files that are written in FV-UNS (FIELDVIEW Unstructured) file format.
- **Version**:
  - *Default Value:* 3.0
  - The value selected details which version of the format should be used when writing the grid and results to files for use within FIELDVIEW. It is recommended that you use the most recent version of the file format supported by FIELDVIEW installed. Refer to the FIELDVIEW documentation for this information.
- **Split Grid and Results Format**:
  - *Default Value:* Selected
  - When selected, the grid from the ANSYS CFX results file is written to one file and the results to a second or subsequent files. If not selected, a single combined file is written containing both the grid and the results.

    > **Note**
    >
    > - This option is not available when the version is less than 2.7.
    > - If the number of variables that are to be written to the file exceeds 200, the results will be written to more than one results file. Refer to FIELDVIEW 9 and later to learn how to load multiple files into FIELDVIEW.
- **Format**:
  - *Default Value:* Unstructured Binary
  - This option enables a choice of whether the FV-UNS are written as in binary or ASCII format. The ASCII format is human-readable but larger than the binary format. It is therefore recommended by ANSYS and FIELDVIEW that binary format files be written in most cases.
- **FV-REG Version**:
  - *Default Value:* 2.0
  - The version specified here details the version of the format used when writing the region file for use in FIELDVIEW. It is recommended that you specify the highest version that is supported by the version of FIELDVIEW installed.

Exported Files

The file set for this export option consists of several files, depending on the format options selected.

If the output format is selected to be Split Grid and Results Format, more than one file will be generated with the grid in one file and results in at least one other. Optionally, a region file will also be generated:

File Name	File Type
<basename>_grid.fv	FIELDVIEW Grid File
<basename>_results.fv	FIELDVIEW Results File
<basename>_region.fv	FIELDVIEW Region File

If **Split Grid and Results Format** (see Available Options (p. 138)) is not selected, then at least one combined grid and results file will be generated:

File Name	File Type
<basename>.fv	FIELDVIEW Combined Grid and Results File
<basename>_grid.fv.fvreg	FIELDVIEW Region File

Optionally, if particle tracks have been written in the ANSYS CFX results file, then these will be written to one or more FIELDVIEW particle track files:

File Name	File Type
<basename>_n.fv	FIELDVIEW Particle Track File

ANSYS CFX writes a subset of record types that are available in the full FIELDVIEW file formats. The documentation supplied with FIELDVIEW describes all the record types that can be read by different versions of FIELDVIEW.

Reading Exported Files in FIELDVIEW

Files can be read into EnSight 5, 6, 7, and 8 as required:

- FIELDVIEW Versions 10.1 and Later (p. 139)
- FIELDVIEW Versions 9 and 10 (p. 140)
- FIELDVIEW Versions 6, 7, 8 (p. 140)

FIELDVIEW Versions 10.1 and Later

If the exported file is split into more than one file, it is necessary to follow a procedure similar to the following to read each file in FIELDVIEW:

1. Select **File > Data Input > ANSYS-CFX[FV-UNS Export]…** .
2. On the **ANSYS-CFX[FV-UNS Export]** form, ensure that the **INPUT MODE** is `Replace`.
3. If the file written is in Split Grid and Results format:
    1. Click **Read Grid or Combined Data**.
    2. Select the grid file from the file popup and click **OK**.
    3. Click **Read Results Data**.
    4. Select the first results file and click **OK**.
    5. If further grid or results files are to be loaded, change the **Input Mode** to `Append` on the **ANSYS-CFX[FV-UNS Export]** form and repeat from step “a” until all grid or results files have been loaded.
4. If the file is a combined file:
    1. Click **Read Grid or Combined Data**.
    2. Select the combined file from the file popup and click **OK**.

FIELDVIEW Versions 9 and 10

If the exported file is split into more than one file, it is necessary to follow a procedure similar to the following to read each file in FIELDVIEW:

1. Select **File > Data Input > Unstructured**.
2. On the **FV Unstructured** form ensure that the **INPUT MODE** is `Replace`.
3. If the file written is in Split Grid and Results format:
    1. Click **Read Grid or Combined Data**.
    2. Select the grid file from the file popup and click **OK**.
    3. Click **Read Results Data**.
    4. Select the first results file and click **OK**.
    5. If further grid or results files are to be loaded, change the input mode to Append on the **FV Unstructured** form and repeat from step “a” until all grid or results files have been loaded.
4. If the file is a combined file:
    1. Click **Read Grid or Combined Data**.
    2. Select the combined file from the file popup and click **OK**.

FIELDVIEW Versions 6, 7, 8

The following procedure enables importing results into FIELDVIEW Version 6, 7 and 8.

> **Note**
>
> FIELDVIEW reads the version number from within the input file; however, some file formats that can be generated by ANSYS CFX cannot be read by all version of FIELDVIEW. Refer to FIELDVIEW documentation for exact details of which file formats can be processed by FIELDVIEW.

1. Select **Data Files** > **Unstructured Data Input** from the menu bar. In **File Selection**, select the file created using cfx5export and then click **OK**. In **Function Subset Selection**, select **All** and click **OK**. Click **Exit** in **Unstructured Data Input**.
2. The results can now be analyzed as required.

> **Note**
>
> The Region file written by ANSYS CFX to these versions of FIELDVIEW may have to be modified to ensure correct axes of rotation and rotational velocities in FIELDVIEW.
>
> If the ANSYS CFX file contains multiple rotation axes, it is not possible to write a single FIELDVIEW file that can be correctly used in FIELDVIEW. In this case, you must write the file in “Split Grid and Results Format”. For more details about reading FIELDVIEW Unstructured files, see the FIELDVIEW documentation.

EnSight

EnSight is a suite of tools for engineering and scientific simulation.

> **Note**
>
> To configure this option, select **Tools** > **Export**. The dialog box uses numerous common export options. For details, see Generic Export Options (p. 132).

This section contains:

Available Options

The following options are available when writing files for use in EnSight:

- All options specified in Generic Export Options (p. 132)
- **EnSight Version**:
  - *Default Value:* Gold
  - The selection specifies the EnSight file format version to which the output will adhere. It is recommended that Gold be selected wherever possible. Refer to the EnSight documentation for differences between the different formats.
- **Format**:
  - *Default Value:* Unstructured Binary
  - This option enables you to choose whether the EnSight data files are written as in binary or ASCII format. The ASCII format is human-readable but larger than the binary format. It is therefore recommended by ANSYS that binary format files are written in most cases.

Export Files

The file set for this export option consists of the following files:

File Name	File Type
<basename>.geom	EnSight Geometry File
<basename>.results	EnSight Results File (version 5 only)
<basename>.s01 (etc.)	EnSight Variable Files for scalar variable
<basename>.v01 (etc.)	EnSight Variable Files for vector variables
<basename>.case	EnSight case files (EnSight 6 and later).

Note

- Each subdomain and boundary condition is exported as one EnSight part.
- CFX variable aliases have to be modified (by removing spaces and special characters and by limiting the name length) to create valid EnSight variable names. In some cases the names that result may not be human recognizable; in all cases the mapping from the name used in ANSYS CFX to that written to the EnSight file is displayed in the progress window of the Solver Manager.

ANSYS CFX uses a subset of record types that are available in the full EnSight file formats. The documentation supplied with EnSight describes all the record types that can be used.

Reading Exported Files into EnSight

Files can be read into EnSight 5, 6, 7, and 8 as required.

- EnSight 8.2 and Later (p. 141)
- EnSight 6, 7, and 8.0 (p. 142)
- EnSight 5 (p. 142)

EnSight 8.2 and Later

The following procedure enables importing results into EnSight 8.2 and later. For more details about importing, see the EnSight documentation.

1. Select **File** > **Open**.
2. Select the **Format** as `Case`.
3. Choose the case file that has been exported. All files produced by CFX-Solver Manager are automatically loaded.

EnSight 6, 7, and 8.0

The following procedure enables importing results into EnSight 6, 7, or 8.0. Results files may need modification, depending on how they are to be used with EnSight. For more details about importing, see the EnSight documentation.

1. Select **File** > **Data Reader**.
2. Select the **Format** as `Case`.
3. Choose the case file that has been exported. All files produced by CFX-Solver Manager are automatically loaded.

EnSight 5

The following procedure enables importing results into EnSight 5. Results files may need modification, depending on how they are to be used with EnSight. For more details about importing, see the EnSight documentation.

1. Select **File** > **Data (Reader)**. In File Selection, click the name of the geometry file created using cfx5export, and then click on **(Set) Geometry**.
2. Click on the name of the EnSight results file. Click **(Set) Result** and then click **Okay**.
3. In **Data Part Loader**, click **Load All**, and then click **Close**.
4. The results can now be analyzed as required.

Custom User Export

Files can be exported to a custom format from CFX-Solver Manager. To do so, a custom export program must be created. For information on all the ways of using a custom export program, see Using a Customized Export Program (p. 147). For details on creating a custom export program, see Creating a Customized Export Program (p. 37).

Export File

This option specifies the destination file to which the grid and results will be written. This file name may be altered in such a way that multiple files may be written with transient data or if the Output Format requires multiple files to be written.

Output Only Boundary Geometry and Results

Availability: All Standard Formats (CGNS, MSC.Patran, FIELDVIEW and EnSight). Available only when **Geometry Output Only** is not selected.

Default Value: Not selected.

If selected, only the data for nodes on the boundaries is output. This can be used, for example, to enable results from CFD calculations to provide boundary conditions for other analysis in other packages.

Mesh Options: Use Initial Mesh for Rotating Domains

Availability: All Standard Formats (CGNS, MSC.Patran, FIELDVIEW and EnSight)

Default Value: Not selected.

By default, the mesh for any rotating domain is rotated into the correct position for the timestep when exporting, as long as the angular velocity is specified as a single value (i.e. not using the CFX Expression Language). You can choose to always write the mesh as it was positioned at the initial timestep by selecting the option **Use Initial Mesh for Rotating Domains**.

If the angular velocity of the rotating domain is specified in terms of an expression, then the mesh is always exported in its initial position and never rotated, and the **Use Initial Mesh for Rotating Domains** option is ignored.

The calculation of the correct rotated position assumes that the initial run was started from a time value of 0 [s], and that the simulation time is continuous from 0 [s] through every run and restart (i.e. the Initial Time is not reset by using the Value option). If this is not the case, then you should select **Use Initial Mesh for Rotating Domains** option to write the mesh as it was positioned at the initial timestep, to avoid incorrectly rotated meshes.

Results Options: Output Level

Availability: All Standard Formats (CGNS, MSC.Patran, FIELDVIEW and EnSight)

Default Value: 1.

The output level selects a predefined subset of variables to write to the destination results file. Each variable is given a “user level” by CFX-Solver. For details, see List of Field Variables (p. 162).

An output level of:

- 1 writes a small subset of basic variables such as velocity and pressure, which have user level 1.
- 2 writes all variables that are of user levels 1 and 2.
- 3 writes all variables stored in the CFX results file.

Results Options: Include Variables Only Found on Boundaries

Availability: All Standard Formats (CGNS, MSC.Patran, FIELDVIEW and EnSight)

Default Value: Not selected.

If selected, the variables that exist only on boundaries (such as y^+) are written in addition to those variables which have values in the interior of the domain(s).

Results Options: Use Corrected Boundary Node Data

Availability: All Standard Formats (CGNS, MSC.Patran, FIELDVIEW and EnSight)

Default Value: Selected

The values of some variables on the boundary nodes (that is, on the edges of the geometry) are not precisely equal to the specified boundary conditions when CFX-Solver finishes calculations.

For instance, the value of velocity on a node on the wall will not be precisely zero, and the value of temperature on an inlet may not be precisely the specified inlet temperature. For visualization purposes, it can be more helpful if the nodes at the boundary contain the specified boundary conditions and so **Use Corrected Data on Boundary Nodes** should be selected for these cases.

Corrected boundary node values are obtained by taking the results produced by CFX-Solver (called “conservative values”) and overwriting the results on the boundary nodes with the values specified by the boundary conditions set up in CFX-Pre. This ensures, for example, that velocity is displayed as zero on no-slip walls and is equal to the specified inlet velocity on an inlet. Using corrected boundary node values is equivalent to selecting conservative variables as described in Hybrid and Conservative Variable Values (p. 161) in the ANSYS CFX Reference Guide..

Running cfx5export from the Command Line

In the following code, `[]` denotes an optional argument, and `<>` and `#` denote that substitution of a suitable value is required. All other arguments are keywords, some of which may have a short form.

> **Note**
>
> `-cgns`, `-ensight`, `-fieldview`, `-patran`, or `-custom` should be the first option on the command line.

- `-cgns` creates files suitable for an application that supports the CGNS format.
- `-ensight` creates files suitable for input into the EnSight post-processor.
- `-patran` creates files suitable for input into the MSC/PATRAN post-processor.
- `-fieldview` creates files suitable for input into the FIELDVIEW post-processor.
- `-custom` should be used if there is a need for custom-defined export formats. For details, see Running a Customized Export Program using cfx5export from the Command Line (p. 148).

For all standard options (`-cgns`, `-patran`, `-fieldview`, and `-ensight`), the source ANSYS CFX file `<file>` is the same file as would be specified under **Tools** > **Export** > **Results File** in CFX-Solver Manager. The optional

file entered as `-name <file>` is the filename used as the basename for the files generated as part of the export process (this is equivalent to what is specified under **Tools** > **Export** > **Export File**).

Running cfx5export

To run cfx5export from the command line, type one of the following commands into a UNIX terminal or a suitable Windows command line and press **Return** or **Enter**. For details, see Command Line (p. 3) in the ANSYS CFX Reference Guide.

```
cfx5export -ansysfsi [-domain <number>] [-help] [-name <file>] \
  [-offset-flow <number>] [-offset-val <number>] \
  [-scale-flow <number>] [-scale-val <number>] \
  [-summary] [-timestep <number>] [-verbose] \
  [-eltype <element type>] [-regions <region1><,region2>...] <file>
```

or

```
cfx5export -cgns [-boundary] [-corrected] [-C] \
 [-domain <number>] [-geometry] [-help] [-name <file>] \
 [-summary] [-timestep <number>] [-user <level>] [-norotate] \
 [-boundaries-as-nodes|-boundaries-as-faces] [-verbose] <file>
```

or

```
cfx5export -ensight [-5|-6|-7] [-absolute] [-ascii|-binary] \
 [-corrected] [-domain <number>] [-help] [-include] [-long] \
 [-name <file>] [-summary] [-timestep <number>] [-user <level>] \
 [-verbose] <file>
```

or

```
cfx5export -fieldview [-UNS[2.4|2.5|2.6|2.7|3.0]] [-REG[1.0|2.0]] \
 [-absolute] [-ascii] [-corrected] [-domain <number>] [-noparticles] \
 [-include] [-name <file>] [-summary] [-timestep <number>] \
 [-user <level>] [-norotate] [-verbose] <file>
```

or

```
cfx5export -patran [-absolute] [-boundary] [-corrected] [-domain <number>] \
 [-geometry] [-help] [-include] [-name <file>] [-nodes] [-summary] \
 [-timestep <number>] [-user <level>] [-norotate] [-verbose] <file>
```

or

```
cfx5export [-exec <executable>] -custom [<options>]
```

Keyword Descriptions

A basic description of the keywords is given below.

Argument	Alternative form	Usage
`-5`		Write an EnSight Version 5 case file instead of an EnSight Version 7 results file.
`-6`		Write an EnSight version 6 file instead of an EnSight version 7 file.
`-7`		Write an EnSight version 7 file (default)

Argument	Alternative form	Usage
-ascii	-s	Write the output file in ASCII format, rather than unstructured binary format.
-binary		Write the output file in unstructured binary format.
-boundary	-b	Boundary data only. Using this argument corresponds to selecting the option **Boundary Data Output Only** when using cfx5export from CFX-Solver Manager.
-boundaries-as-faces		Write boundary conditions as collections of faces rather than nodes.
-boundaries-as-nodes		Write boundary conditions as collections of nodes rather than faces (default).
-cgns		Export data in the CGNS format
-cgns-names		Map CFX variable names to CGNS variable names (where possible).
-corrected	-c	Use corrected boundary node values. This corresponds to enabling **Use Corrected Boundary Node Data** when using **Tools > Export** from CFX-Solver Manager.
-one-file-for-transient-cases	-C	Save all timesteps into one CGNS file.
-custom <options>		Use custom export program defined by the CFX_EXPORT_EXEC variable. If this option is specified no further argument checking is done and all remaining options on the command line are passed straight to the export program. For details, see Generic Export Options (p. 132).
-domain <number>	-d <number>	Specifies the domain of interest. If <number> is non-zero, cfx5export will export just the given domain. If <number> is zero, the data is combined and exported as a single domain. The default value is 0.
-ensight		Export data for use with EnSight.
-exec <executable>		An alternative way of specifying the custom export executable dynamically, without having to create a cfx5rc file. Note that this parameter must appear before the -custom switch.
-fieldview	-fv	Export data for use with FIELDVIEW.
-geometry	-g	Geometry data only (no results). Using this option corresponds to choosing **Geometry Output Only (Neutral File)** when using cfx5export from CFX-Solver Manager.
-help	-h	Print this information.
-include	-i	Include boundary node only data. If you specify this option, then variables such as y^+ (which have meaningful values only on the boundary nodes) will be exported.
-long	-l	Use long variable names.

Argument	Alternative form	Usage
-one-file-for-transient-cases		Write all transient data to a single CGNS file using BaseIterative_t and ZoneIterative_t nodes.
-name <base>	-output <base> -out <base>	Set the basename for the output files to <base> instead of the name of the input file. If you do not use this option, the exported files will be written in the same directory as the input file. You need to take care when selecting this name to avoid your CFX results file being overwritten.
-nodes	-n	Use nodes when exporting packet 21 data (groups).
-noparticles		Do not write particle track files.
-norotate		Do not rotate the grid to its true position. For details, see Mesh Options: Use Initial Mesh for Rotating Domains (p. 142).
-patran		Export data for use with MSC/PATRAN.
-REG <version>		Specifies the version used when writing the FIELDVIEW region file.
-split-grid-and-results		Use the FIELDVIEW Split Grid and Results file format when writing grid and results files.
-summary	-f	Displays a summary of the domains and timesteps contained in the CFX results file.
-timestep <number>	-time <number> -t <number>	If the <number> is non-zero, data for the given timestep in a transient run is exported. If <number> is -1, data from all timesteps is exported. If the -timestep switch is not given, data from the last timestep will be exported.
-UNS<version>		Specifies the file format version when writing FIELDVIEW Grid and Results files.
-user <level>	-u <level>	User level of interest: <level> should be a number 1, 2 or 3, and this corresponds to selecting Output Level 1, 2 or 3 when using cfx5export from CFX-Solver Manager.
-verbose	-v	Specifying this option may result in additional output being sent to the standard output file (normally the screen).
<file>		Name of CFX results file from which data is to be exported. The names of the file(s) created depends on the format being written. CGNS files will be written to <file>.cgns EnSight geometry will be written to <file>geom, the results file to <file>.res, and the variables to <file>.s## or <file>.v## where ## is the variable number and s indicates a scalar and v a vector unless the **-name** option is used. Patran geometry will be written to <file>.out as a Neutral file, the results template to <file>.res_tmpl, and the nodal results to <basename>.nod unless the **name** option is used.

Argument	Alternative form	Usage
		Fieldview geometry will be written to `<file>.fv` unless the **-name** option is used. Note: The files will be created in the same directory as the original files, not necessarily in the current working directory.

Exporting a Transient Results File

If you have completed a transient run, you have several options for exporting the results. You can elect to:

- Export the results file, which contains the solution to the final timestep.
- Export one or more of the preliminary timestep solutions.
- Export all of the timesteps.

You can use either CFX-Solver Manager or the cfx5export script to export a transient file.

- Generic Export Options (p. 132)
- Running cfx5export from the Command Line (p. 143)

File Format

If you elect to export the results file, which contains the solution to the final timestep, the format will be:

```
filename.ext
```

where `.ext` is based on the format being written.

However, if you elect to export a different timestep, then the export file will have the following format:

```
filename_t#.ext
```

where # is the value of the timestep exported.

> **Note**
>
> EnSight transient files will be sequentially numbered, regardless of the timestep value. For example, if timesteps of 1s, 5s, and 7s are exported, they will have the filenames `filename_t1.ext`, `filename_t2.ext` and `filename_t3.ext`.

Exporting Particle Tracking Data

Export of particle tracking data is currently supported only to FIELDVIEW. Export of particle tracking data to any other format is not supported in the current release of ANSYS CFX.

Using a Customized Export Program

There are multiple ways in which an export program can be used once it has been compiled.

- Using a Customized Export Program from CFX-Solver Manager (p. 147)
- Using a Customized Export Program from the Command Line (p. 148)

Using a Customized Export Program from CFX-Solver Manager

Open the Export dialog box by selecting **Export** from the **Tools** menu of CFX-Solver Manager.

1. Set **Output Format** to `Custom User Export`.
2. Supply the path to the compiled program in **Export Executable**.

3. Enter any associated options required to run the customized program (such as would be entered at the command line) in **Custom Export Setting**.
4. The specified custom export program runs with the associated arguments when you click **Export**.

Using a Customized Export Program from the Command Line

You may run your program directly from the command line started from the launcher (**Tools > Command Line**). Such a command line will have the environment variables and path set appropriately.

> **Important**
>
> It is important to run the command line in the CFX environment. For details, see Command Line (p. 3) in the ANSYS CFX Reference Guide.

Running a Customized Export Program Directly from the Command Line

For the purposes of describing what you can do, assume that the executable file is called `myexport` and is contained in the directory `/home/smith/export/` (UNIX) or `c:\smith\export` (Windows). You want to specify the results file called `file.res` in the same directory and make the basename `example`.

UNIX

To run the program directly on a UNIX system, open a command line started from the launcher (**Tools > Command Line**) and call your program using:

```
./myexport file.res example
```

assuming that the current directory is `/home/smith/export/`.

Windows

On a Windows machine, run the program by opening a command line started from the launcher (**Tools > Command Line**), changing directory to `c:\smith\export`, and typing:

```
myexport file.res example
```

> **Note**
>
> Just double-clicking on the name of the program in the Windows Explorer, or using the **Execute** `myexport.exe` option in Microsoft Developer Studio does not readily give you the option to enter command-line arguments.

Running a Customized Export Program using cfx5export from the Command Line

Your executable can be run directly from the command line by using the cfx5export command. This enables you to issue an export command without specifying the location of the executable each time; an environment variable remembers the location of the custom export executable.

For the purposes of describing the procedure, assume the executable file is called myexport and is contained in the directory `/home/smith/export/` (UNIX) or `c:\smith\export` (Windows). Specify the results file called `file.res` in the same directory and make the basename example.

To run the program using the `cfx5export -custom` command, add the following line to the `.cfx5rc` file:

```
CFX_EXPORT_EXEC="<executable_path>"
```

where `<executable_path>` is the full path and name of the executable (for example, `/home/smith/export/myexport` or `c:\smith\export\myexport` as appropriate). For details, see Resources Set in cfx5rc Files (p. 38) in the ANSYS CFX Introduction.

The cfx5export command can be used with the custom argument for the given example. Type:

```
cfx5export -custom file.res example
```

into a UNIX terminal or a suitable Windows command prompt and press **Return** or **Enter**. For details, see Command Line (p. 3) in the ANSYS CFX Reference Guide.

Index

Symbols

A

B

C

D

E

F

G

H

I

L

M

N

O

P

R

S

T

U

V

W

Z