

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 14.0
November 2011

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

Copyright and Trademark Information

© 2011 SAS IP, Inc. All rights reserved. Unauthorized use, distribution or duplication is prohibited.

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

Disclaimer Notice

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

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

U.S. Government Rights

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

Third-Party Software

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

Published in the U.S.A.

Table of Contents

1. CFX-Solver Manager Basics	1
1.1. Starting CFX-Solver Manager	1
1.2. Working with the CFX-Solver Manager Interface	1
1.2.1. Workspace Selector	2
1.2.2. Convergence History Plots and User Point Plots	2
1.2.2.1. Printing an Image of the Convergence History	3
1.2.2.2. Saving a Picture of the Convergence History	4
1.2.2.3. Exporting Plot Data	4
1.2.2.4. Exporting Monitor Data from the Command Line	4
1.2.3. Text Output Window	5
1.2.3.1. Searching for Text	5
1.2.3.2. Saving the Text to File	5
1.2.4. Multi-Configuration Run History Page	6
1.3. Customizing CFX-Solver Manager	7
2. Working with Solver Manager	9
2.1. Solver Run Overview	9
2.2. The Define Run Dialog Box	10
2.2.1. Solver Input File Name	10
2.2.2. Edit Configuration	10
2.2.3. Run Definition Tab	11
2.2.4. MultiField Tab	12
2.2.5. Partitioner Tab	13
2.2.6. Solver Tab	16
2.2.7. Interpolator Tab	16
2.2.8. Configuring Memory for the CFX-Solver	17
2.3. Run Output Results	17
2.4. Parallel Run	19
2.4.1. Overview	19
2.4.2. General Procedure	19
2.4.3. Configuring a Parallel Run	19
2.4.3.1. Local Parallel Setup	20
2.4.3.2. Distributed Parallel Setup	20
2.4.3.2.1. Overview	20
2.4.3.2.2. Selecting Parallel Hosts	20
2.4.3.2.3. Configuring a Host	21
2.5. Restarting a Run	21
2.5.1. Restart Procedure	21
2.5.2. Restart Details	21
2.5.2.1. Runs Using Mesh Adaption	22
2.5.2.2. Runs After Physical Model or Solver Parameter Changes	22
2.5.2.3. Runs After Topology or Mesh Changes	22
2.5.2.4. Multi-Configuration Simulations	23
2.6. ANSYS Multi-field Run	23
2.6.1. Overview	24
2.6.2. General Procedure	24
2.6.2.1. Starting ANSYS CFX and ANSYS Separately	25
2.6.2.2. Process ANSYS Input File Only	25
2.6.2.3. Directory Structure	26
2.6.2.4. Starting an ANSYS Multi-field Run from the Command Line	26
2.6.3. Monitoring a Run in Progress	26

2.6.4. Monitoring a Completed Run	27
2.6.5. ANSYS Multi-field Residual Plotting	27
2.6.6. Processing the ANSYS Input File	27
2.6.6.1. When the ANSYS Input File Needs Processing	27
2.6.6.2. The Processing Step	28
2.6.7. Restart Procedure for ANSYS Multi-field Runs	29
2.6.7.1. Restart from End of Previous Run	29
2.6.7.2. Restart from Intermediate Point During Previous Run	29
2.6.7.3. Changing CFX Physics or ANSYS Multi-field Settings	31
2.6.7.4. Changing ANSYS Physics	31
2.6.7.5. Restart Limitations	32
3. CFX-Solver Files	33
3.1. Files Used by the CFX-Solver	33
3.2. Files Generated by the CFX-Solver	35
3.2.1. CFX-Solver Output File	36
3.2.1.1. Header	36
3.2.1.2. CFX Command Language for the Run	36
3.2.1.3. CFD Job Information	40
3.2.1.4. Memory Allocated for the Run	40
3.2.1.5. Mesh Statistics	41
3.2.1.6. Initial Average Scales	42
3.2.1.7. Checking for Isolated Fluid Regions	42
3.2.1.8. Solved Equations	43
3.2.1.9. Convergence History	43
3.2.1.10. Computed Model Constants	45
3.2.1.11. Termination and Interrupt Condition Summary	45
3.2.1.12. Global Conservation Statistics	45
3.2.1.13. Calculated Wall Forces and Moments	47
3.2.1.14. Maximum Residual Statistics	49
3.2.1.15. False Transient Information	49
3.2.1.16. Final Average Scales	50
3.2.1.17. Variable Range Information	50
3.2.1.18. CPU Requirements	50
3.2.1.19. Job Information	51
3.2.2. CFX-Solver Output File (Transient Runs)	51
3.2.2.1. Convergence History	51
3.2.3. CFX-Solver Output File (Transient Blade Row Runs)	52
3.2.3.1. Post-processing Information	52
3.2.3.2. Fourier Transformation Information	53
3.2.3.3. Stability Information (Time Transformation Runs)	53
3.2.4. CFX-Solver Output File (Interpolation Runs)	54
3.2.5. CFX-Solver Output File (Parallel Runs)	55
3.2.5.1. Partitioning Information	55
3.2.5.2. Job Information	56
3.2.5.3. Host Information	56
3.2.5.4. Memory Usage Information	56
3.2.5.5. Completed Job Information	57
3.2.6. CFX-Solver Output File (Mesh Adaption Runs)	57
3.2.7. CFX-Solver Output File (Remeshing Runs)	58
3.2.8. CFX-Solver Output File (Conjugate Heat Transfer Runs)	58
3.2.8.1. Thermal Energy Flow Through a Solid Boundary	58
3.2.8.2. Thermal Energy Flow Between the Fluid and Solid within a Porous Domain	59

3.2.9. CFX-Solver Output File (GGI Runs)	59
3.2.10. CFX-Solver Output File (Combustion Runs)	60
3.2.10.1. Multicomponent Specific Enthalpy Diffusion (MCF)	60
3.2.10.2. Single Step Reactions Heat Release	61
3.2.10.3. Stoichiometric Mixture Fraction	61
3.2.10.4. Hydrocarbon Fuel Model: Proximate / Ultimate Analysis	61
3.2.11. CFX-Solver Output File (Particle Runs)	62
3.2.11.1. Particle Transport Equations	62
3.2.11.2. Particle Fate Diagnostics	63
3.2.11.2.1. Absorbed by Porous Media	63
3.2.11.2.2. Continue from Last Time Step (Transient Only)	63
3.2.11.2.3. Collected on Walls	63
3.2.11.2.4. Entered Domain	63
3.2.11.2.5. Exceeded Distance Limit	63
3.2.11.2.6. Exceeded Integration Limit	64
3.2.11.2.7. Exceeded Time Limit	64
3.2.11.2.8. Fell Below Minimum Diameter	64
3.2.11.2.9. Integration Error	64
3.2.11.2.10. Left Domain	64
3.2.11.2.11. Sliding along Walls	64
3.2.11.2.12. Waiting for Next Time Step (Transient only)	64
3.2.11.3. Transient Particle Diagnostics	65
3.2.11.4. Particle Convergence History	65
3.2.11.5. Integrated Particle Flows	65
3.2.11.6. CPU Requirements of Numerical Solution	66
3.2.12. CFX-Solver Output File (ANSYS Multi-field Runs)	66
3.2.13. CFX-Solver Output File (Radiation Runs)	67
3.2.13.1. Convergence History	67
3.2.13.1.1. P1 Model	67
3.2.13.1.2. Discrete Transfer Model	67
3.2.13.1.3. Monte Carlo Model	68
3.2.13.2. Summary Fluxes	68
3.2.14. CFX-Solver Output File (Rigid Body Runs)	69
3.2.15. CFX-Solver Results File	69
3.2.15.1. CFD-Post	70
3.2.15.2. CFX-Solver	70
3.2.16. CFX Radiation File	70
3.2.16.1. CFX Radiation File Contents	71
3.2.16.2. Volume Information	71
3.2.16.3. Surface Information	72
3.2.17. CFX Partition File	72
3.2.18. Additional Files for ANSYS Multi-field Runs	73
3.2.19. CFX Multi-Configuration Output File	73
3.2.19.1. Header	73
3.2.19.2. CFX Command Language for the Run	73
3.2.19.3. Simulation Execution Summary	76
3.2.19.4. Simulation Termination Condition Summary	77
3.2.20. CFX Multi-Configuration Results File	78
4. Residual Plotting	79
4.1. Equation Residual	79
4.2. Convergence Results and RMS	79
4.3. Transient Residual Plotting	80

4.4. Residual Plotting for ANSYS Multi-field Runs	80
4.4.1. ANSYS Field Solver Plots	81
4.4.2. ANSYS Interface Loads Plots	81
5. Editing CFX-Solver Input Files	83
5.1. Workflow Overview	83
5.2. Command File Editor Overview	83
5.3. Menus in the Command File Editor	84
5.3.1. File Menu	84
5.3.1.1. Save Command	84
5.3.1.2. Validate Command	84
5.3.1.3. Exit Command	84
5.3.2. Edit Menu	84
5.3.2.1. Add Parameter Command	85
5.3.2.2. Edit Parameter Command	85
5.3.2.2.1. Expanding Categories	85
5.3.2.2.2. Editing Entries	86
5.3.2.3. Delete Parameter Command	86
5.3.2.4. Add Expert Parameter Section Command	86
5.3.2.5. Find Command	87
5.3.2.6. Find Next Command	87
5.3.3. Help Menu	87
5.3.3.1. On CFX Solver File Editor Command	87
5.3.3.2. On Expert Parameters Command	87
5.4. Command File Editor Rules	87
5.5. Command File Editor Appearance	87
5.6. Editing the Command Language (CCL) File	88
5.7. Command Language File Rules	88
5.8. RULES and VARIABLES Files	89
5.8.1. VARIABLES File	89
5.8.2. RULES File	90
5.8.2.1. RULES	91
5.8.2.2. SINGLETON	91
5.8.2.3. OBJECT	91
5.8.2.4. PARAMETER	92
6. CFX-Solver Manager File Menu	93
7. CFX-Solver Manager Edit Menu	95
7.1. CFX-Solver Manager Options	95
7.2. Common Options	96
7.2.1. Appearance	96
7.2.2. Viewer Setup	97
7.2.2.1. Double Buffering	97
7.2.2.2. Unlimited Zoom	97
7.2.2.3. Mouse Mapping	97
7.2.3. Units	97
8. CFX-Solver Manager Workspace Menu	99
8.1. Workspace Properties Command	99
8.1.1. General Settings Tab	99
8.1.2. Monitors Tab	100
8.1.2.1. Plot Monitor	100
8.1.2.2. Residual Monitor	100
8.1.2.3. Text Monitor	100
8.1.2.4. Filter Selector	100

8.1.3. Creating Monitors	100
8.1.4. Modifying Monitors	101
8.1.4.1. Deleting Monitors	101
8.1.4.2. Monitor Properties	101
8.1.4.2.1. General Settings Tab	101
8.1.4.2.2. Range Settings Tab	102
8.1.4.2.3. Plot Lines Tab	103
8.1.4.2.4. CFX Plot Line Variables	103
8.1.4.2.4.1. ACCUMULATED TIMESTEP	104
8.1.4.2.4.2. BALANCES	104
8.1.4.2.4.3. FLOW	104
8.1.4.2.4.4. FORCE	104
8.1.4.2.4.5. NEG ACCUMULATION	104
8.1.4.2.4.6. RESIDUAL	104
8.1.4.2.4.7. SOURCE	104
8.1.4.2.4.8. TIMESTEP	104
8.1.4.2.4.9. USER	105
8.1.4.2.4.9.1. Configuring Plot Lines	105
8.1.4.2.4.10. ANSYS GST and NLH Variables	105
8.1.4.2.4.11. ANSYS	105
8.1.5. Plotting by a Specific Variable	106
8.1.6. Global Plot Settings Tab	106
8.2. New Monitor Command	107
8.3. Stop Current Run Command	107
8.3.1. Stopping Runs Using Mesh Adaption	107
8.3.2. Stopping Runs Using Remeshing	107
8.3.3. Stopping Runs Using External Solver Couplings	107
8.3.4. Stopping Multi-Configuration Runs	107
8.4. Restart Current Run Command	108
8.5. Backup Run Command	108
8.6. Arrange Workspace Command	108
8.7. Toggle Layout Type Command	108
8.8. Load Layout Command	109
8.9. Save Layout Command	109
8.9.1. Defining a Default Plot Monitor	110
8.10. View RMS Residuals Command	110
8.11. View MAX Residuals Command	110
8.12. Reset to Default Workspace Command	110
8.13. Close Workspace Command	111
9. CFX-Solver Manager Tools Menu	113
9.1. Edit CFX-Solver File Command	113
9.2. Export Command	113
9.3. Export to ANSYS MultiField Command	114
9.4. Interpolate Command	115
9.4.1. Using the Command Line to Interpolate Results	117
9.5. Edit Run In Progress Command	118
9.6. CCL Propagation in Multi-Configuration Simulations	118
9.7. Edit Current Results File Command	119
9.8. Post-Process Results Command	119
9.9. View Environment Command	119
10. CFX-Solver Manager Monitors Menu	121
11. Starting the CFX-Solver from the Command Line	123

11.1. Command-Line Use	123
11.2. Command-Line Options and Keywords for cfx5solve	124
11.3. Command-Line Samples	135
12. CFX-Solver Start Methods	139
13. CPU and Memory Requirements	141
13.1. Tetrahedral Mesh	141
13.2. Special Partitioner, Solver and Interpolator Executables	141
13.3. Turbulence	142
13.4. Energy Models	142
13.5. CHT Regions	143
13.6. Multicomponent Flows	143
13.7. Multiphase Flows	143
13.8. Additional Variables, Wall Distance Variables, and Boundary Distance Variables	144
13.9. Combustion Modeling	144
13.10. Radiation Modeling	144
13.11. GGI Interfaces	144
13.12. Transient Runs	144
13.13. Mesh Deformation	144
13.14. Bidirectional (Two-Way) Couplings with ANSYS Multi-field	145
14. The cfx5control Application	147
15. File Export Utility	149
15.1. Export to ANSYS Multi-field Solver Dialog Box	149
15.1.1. ANSYS Element Type	150
15.1.2. Output Modifiers	150
15.1.2.1. Offset Flow	150
15.1.2.2. Offset Values	150
15.1.2.3. Scale Flow	150
15.1.2.4. Scale Values	150
15.2. Export of Results to Other Formats	150
15.3. Generic Export Options	151
15.3.1. Results File	151
15.3.2. Domain Selection: Name	151
15.3.3. Timestep Selection: Timestep	151
15.3.4. Output Format	151
15.3.4.1. CGNS	152
15.3.4.1.1. CGNS Options	152
15.3.4.1.2. Exported Files	153
15.3.4.1.3. Contents of CGNS Files Written by ANSYS CFX	153
15.3.4.1.3.1. Base (Base_t)	153
15.3.4.1.3.2. Zones (Zone_t)	154
15.3.4.1.3.3. Elements (Elements_t)	154
15.3.4.1.3.4. Boundary Conditions (BC_t)	154
15.3.4.1.3.5. Solution Data (FlowSolution_t)	154
15.3.4.1.3.6. Transient Data	154
15.3.4.1.3.7. ANSYS CFX Connectivity using CGNS for Aerodynamic Noise Analysis	154
15.3.4.1.4. Reading Exported Files into a Program Supporting CGNS	154
15.3.4.2. MSC.Patran	155
15.3.4.2.1. Available Options	155
15.3.4.2.2. Exported Files	155
15.3.4.2.3. Reading Exported Files into MSC.Patran	156
15.3.4.2.4. Exporting Boundary Conditions to MSC.Patran	156
15.3.4.2.5. Example Procedure	157

15.3.4.3. FIELDVIEW	158
15.3.4.3.1. Available Options	158
15.3.4.3.2. Exported Files	159
15.3.4.3.3. Reading Exported Files in FIELDVIEW	160
15.3.4.3.3.1. FIELDVIEW Versions 10.1 and Later	160
15.3.4.3.3.2. FIELDVIEW Versions 9 and 10	160
15.3.4.3.3.3. FIELDVIEW Versions 6, 7, 8	161
15.3.4.4. EnSight	161
15.3.4.4.1. Available Options	161
15.3.4.4.2. Export Files	162
15.3.4.4.3. Reading Exported Files into EnSight	162
15.3.4.4.3.1. EnSight 8.2 and Later	162
15.3.4.4.3.2. EnSight 6, 7, and 8.0	163
15.3.4.4.3.3. EnSight 5	163
15.3.4.5. Custom User Export	163
15.3.5. Export File	163
15.3.6. Output Only Boundary Geometry and Results	163
15.3.7. Mesh Options: Use Initial Mesh for Rotating Domains	164
15.3.8. Results Options: Output Level	164
15.3.9. Results Options: Include Variables Only Found on Boundaries	164
15.3.10. Results Options: Use Corrected Boundary Node Data	164
15.4. Running cfx5export from the Command Line	165
15.4.1. Running cfx5export	165
15.4.2. cfx5export Arguments	166
15.5. Exporting a Transient Results File	169
15.5.1. File Format	169
15.6. Exporting Particle Tracking Data	170
15.7. Using a Customized Export Program	170
15.7.1. Using a Customized Export Program from CFX-Solver Manager	170
15.7.2. Using a Customized Export Program from the Command Line	170
15.7.2.1. Running a Customized Export Program Directly from the Command Line	170
15.7.2.1.1. UNIX	170
15.7.2.1.2. Windows	171
15.7.2.2. Running a Customized Export Program using cfx5export from the Command Line	171
Index	173

Chapter 1: CFX-Solver Manager Basics

CFX-Solver Manager is a graphical user interface that enables 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 *Starting the CFX-Solver from the Command Line* (p. 123)).

This chapter describes:

- 1.1. Starting CFX-Solver Manager
- 1.2. Working with the CFX-Solver Manager Interface
- 1.3. Customizing CFX-Solver Manager

1.1. Starting CFX-Solver Manager

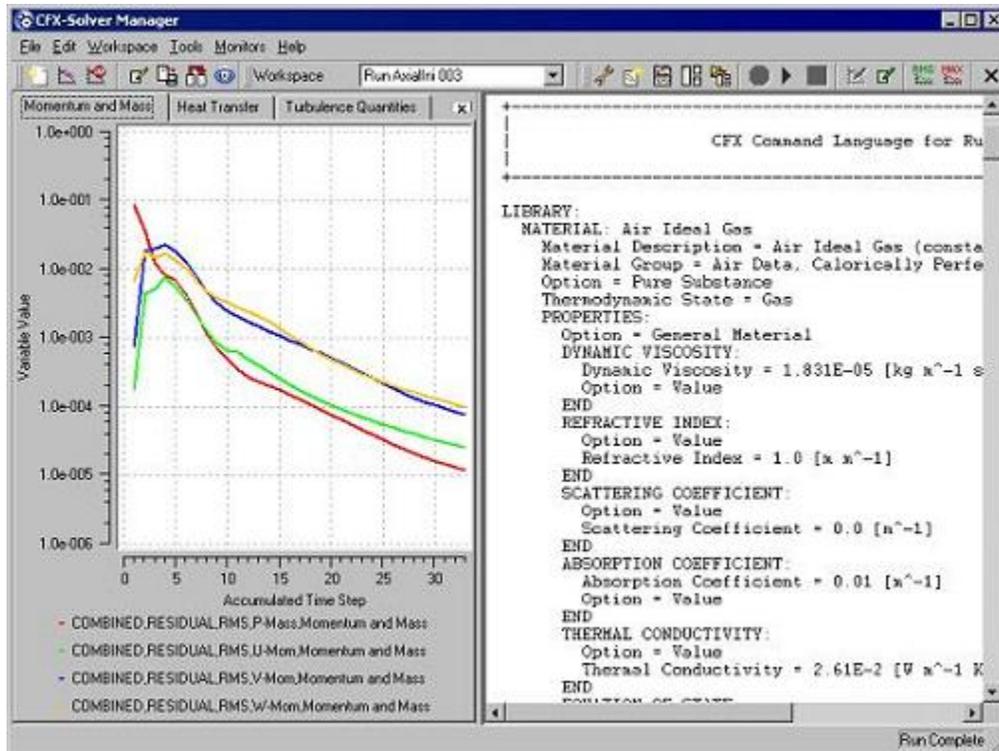
You can start CFX-Solver Manager in different ways:

- Using ANSYS Workbench. For details, see *ANSYS CFX in ANSYS Workbench in the CFX Introduction*.
- Using the ANSYS CFX Launcher. For details, see *Using the ANSYS CFX Launcher in the 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 in the CFX-Pre User's Guide*.

1.2. 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](#) (p. 2)
- [Convergence History Plots and User Point Plots](#) (p. 2)
- [Text Output Window](#) (p. 5)
- [Multi-Configuration Run History Page](#) (p. 6)

1.2.1. 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.

1.2.2. Convergence History Plots and User Point Plots

Convergence history plots and user point plots are shown on monitor tabs. Each monitor tab shows at least one plot of a variable versus time step, where the variable can be an RMS or maximum residual, or a user-defined variable. For example, the **Momentum and Mass** monitor tab shows plots of the RMS/maximum residuals for pressure and the U, V, and W components of momentum. A legend appears below each plot to show the variable associated with each plot line.

For simulations involving rigid body modeling, additional plots specific to rigid bodies are available. For details, see [Monitor Plots related to Rigid Bodies in the CFX-Solver Modeling Guide](#).

For Transient Blade Row simulations involving the Time Transformation method, the flow equations produce results in “computational time,” not “physical time.” By default, plots that have titles that begin with “Time Corrected” are in terms of physical time, and all other plots are in terms of computational time. A plot that uses physical time has the independent axis labeled “Simulation Time.” A plot that uses computational time has the independent axis labeled “Accumulated Time Step.” To switch a plot to use physical time, right-click a plot and select **Monitor Properties** from the shortcut menu, then, in the **Monitor Properties** dialog box, go to the **Range Settings** tab and change **Plot Data by** to *Simulation Time*. To switch a plot to use computational time, change the same setting to *Time Step*. Note that, in a plot that uses physical time, the lines may be offset from each other in terms of simulation time, and only data with positive physical time is displayed. For details on the Time Transformation method, see [Time Transformation Method in the CFX-Solver Theory Guide](#).

You can control which plots are active from the **Monitors** menu.

Some other actions that you can perform:

- Configure/edit a monitor from the [Monitor Properties \(p. 101\)](#) dialog box. To invoke that dialog box, you can do one of the following:
 - Right-click in the plot area of the monitor tab and select **Monitor Properties** from the shortcut menu.
 - Select **Workspace > Workspace Properties** from the main menu, then select a monitor from the **Workspace Properties** dialog box and click *Edit* .
- Click any of the plot lines to view the value at the nearest time step.
- Toggle between showing RMS or maximum values by right-clicking in the monitor tab and selecting **Switch Residual Mode** from the shortcut menu.
- Hide or show any monitor tab from the [CFX-Solver Manager Monitors Menu \(p. 121\)](#). Another way to hide a monitor tab (other than the **Out File** monitor tab) is to right-click in the monitor tab and select **Hide Monitor** from the shortcut menu.
- Delete a monitor tab by right-clicking in the monitor tab and selecting **Delete Monitor** from the shortcut menu.

Other actions that can be performed using the shortcut menu on the monitor tab are described in the following sections:

- [Printing an Image of the Convergence History \(p. 3\)](#)
- [Saving a Picture of the Convergence History \(p. 4\)](#)
- [Exporting Plot Data \(p. 4\)](#)
- [Exporting Monitor Data from the Command Line \(p. 4\)](#)

1.2.2.1. 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**.

The **Print** dialog box appears.

2. Configure the printer as required.
3. Click **Print**.

1.2.2.2. 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 appears.

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**.

1.2.2.3. 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 and select **Export Plot Data**.

The **Export** dialog box appears.

2. Select a location to which to export the data.
3. Under **File name**, enter the name for the file.
4. Under **File Type**, select the format to export.

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

5. Click **Save**.

The data from each plotted variable is exported to the specified file.

Note

Monitor data can also be exported using the command-line application `cfx5mondata`.

Note

When exporting from a time-corrected plot, you can export data (one or more variables) for only one monitor point; each selected plot line variable must belong to the same monitor point. This restriction guarantees that all exported data is written in terms of the same corrected time. You can change which plot line variables are selected by using the **Monitor Properties** dialog box (right-click in the plot area and select **Monitor Properties** from the shortcut menu).

1.2.2.4. 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
```

1.2.3. Text Output Window

The text output window is a monitor tab that 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. The source of the displayed information is an output file from a solver run. For details, see *CFX-Solver Output File* (p. 36).

For an ANSYS Multi-field run, an additional text output window shows text output from the ANSYS solver. In this case, you can switch between viewing the CFX output file and the ANSYS output 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".

You can hide or show the **Out File** monitor tab from the *CFX-Solver Manager Monitors Menu* (p. 121).

Right-click in the **Out File** monitor tab to access a shortcut menu with commands for finding text, selecting text, copying text, saving the text to a file, managing bookmarks, editing the properties of the **Out File** monitor tab, and deleting the **Out File** monitor tab.

The following sections explain some of these actions in more detail:

- [Searching for Text](#) (p. 5)
- [Saving the Text to File](#) (p. 5)

1.2.3.1. 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.

1.2.3.2. Saving the Text to File

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

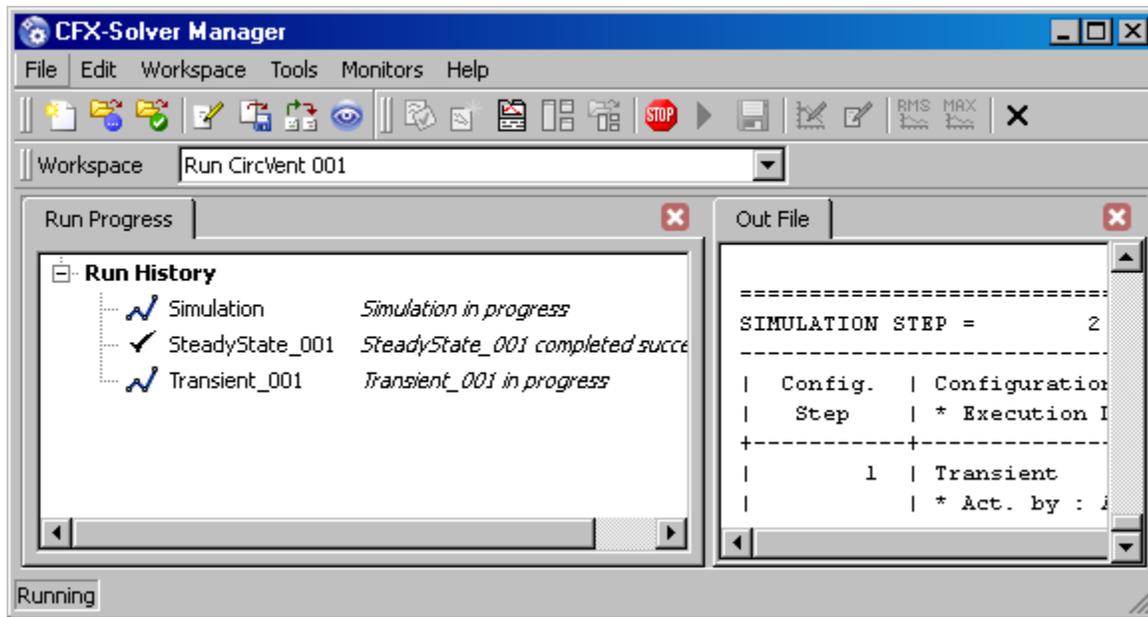
1. Right-click in the text output window.
2. Select **Save As**.
3. Select a file location to save the text file.
4. Enter a file name for the text file.
5. Click **Save**.

1.2.4. 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:

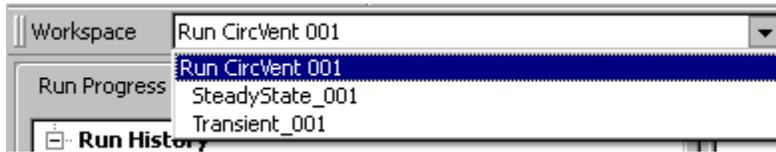
- Open Workspace
 - This switches to the workspace view for the selected simulation step.
- Post Process Results
 - This command is available when running CFX-Solver Manager in stand-alone mode. It launches CFD-Post with the results of the selected simulation step.
- Post Process Results and Shutdown
 - 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.

1.3. 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 placed 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 [CFX-Solver Manager Workspace Menu](#) (p. 99).

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:

- 2.1. Solver Run Overview
- 2.2. The Define Run Dialog Box
- 2.3. Run Output Results
- 2.4. Parallel Run
- 2.5. Restarting a Run
- 2.6. ANSYS Multi-field Run

2.1. 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 enables the coupling of the CFX-Solver with the ANSYS Solver in order to execute cases that 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.

2.2. 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:

- *Solver Input File Name* (p. 10)
- *Edit Configuration* (p. 10)
- *Run Definition Tab* (p. 11)
- *MultiField Tab* (p. 12)
- *Partitioner Tab* (p. 13)
- *Solver Tab* (p. 16)
- *Interpolator Tab* (p. 16)

2.2.1. 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. 123).

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.

2.2.2. 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**.

2.2.3. 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 in the 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 in the 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**. 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 in the CFX-Solver Manager User's Guide](#). The precisions of the partitioner, solver, and interpolator executables can be set individually on the **Partitioner**, **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 in the 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).

- A serial run (the default) requires no additional configuration
- To learn how to configure a parallel run, see [Parallel Run in the CFX-Solver Manager User's Guide](#).

2.2.4. 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 in the CFX-Solver Modeling Guide](#). A description of the different run modes and settings can be found in [ANSYS Multi-field Run in the CFX-Solver Manager User's Guide](#).

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
MultiField Remote Solver Settings				
Host Name			Required	
Host Port				

^aThe 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 in the CFX-Solver Manager User's Guide](#).

^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 `v140/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.

2.2.5. 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 in the CFX-Solver Manager User's Guide](#).

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 in the CFX-Solver Manager User's Guide](#).
4. Optionally, set the precision: under **Executable Settings**, select **Override Default Precision** and choose either **Single** or **Double**. This setting for the partitioner will override the corresponding specification, if set, on the **Run Definition** tab.

For details, see [Double-Precision Executables in the CFX-Solver Manager User's Guide](#).

5. 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 in the CFX-Solver Manager User's Guide](#).
6. 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 in the CFX-Solver Modeling Guide](#). When first running in parallel, it is recommended that **Partition Type** be set to **MeTiS**.
 - [Recursive Coordinate Bisection in the CFX-Solver Modeling Guide](#)
 - [Optimized Recursive Coordinate Bisection in the CFX-Solver Modeling Guide](#)
 - [Directional Recursive Coordinate Bisection in the CFX-Solver Modeling Guide](#)
 - [User Specified Direction in the CFX-Solver Modeling Guide](#)
 - [Simple Assignment in the CFX-Solver Modeling Guide](#)
 - [Radial in the CFX-Solver Modeling Guide](#)
 - [Circumferential in the CFX-Solver Modeling Guide](#)
7. If required, configure the **Partition Weighting** as described below.
 8. 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 in the 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**.

Performance of particle transport calculations may be made worse when using coupled partitioning.

- If required, under **Partitioner Memory**, adjust the memory configuration. For details, see [Configuring Memory for the CFX-Solver in the CFX-Solver Manager User's Guide](#).

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 in the CFX-Solver Manager User's Guide](#).

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.

2.2.6. Solver Tab

1. Select the **Solver** tab.

If this is not available, ensure **Show Advanced Controls** in 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 in the CFX-Solver Manager User's Guide](#).
3. Optionally, set the precision: under **Executable Settings**, select **Override Default Precision** and choose either **Single** or **Double**. This setting for the solver will override the corresponding specification, if set, on the **Run Definition** tab.

For details, see [Double-Precision Executables in the CFX-Solver Manager User's Guide](#).

4. If required, under **Solver Memory**, adjust the memory configuration. For details, see [Configuring Memory for the CFX-Solver in the CFX-Solver Manager User's Guide](#).
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**.

Note

The **Custom Solver Options** (**Custom Executable** and **Solver Arguments**) are available in the CFX-Solver Manager in stand-alone mode.

2.2.7. Interpolator Tab

1. Select the **Interpolator** tab.

If this is not available, ensure **Show Advanced Controls** in the **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 in the CFX-Solver Manager User's Guide](#).
3. Optionally, set the precision: under **Executable Settings**, select **Override Default Precision** and choose either **Single** or **Double**. This setting for the interpolator will override the corresponding specification, if set, on the **Run Definition** tab.

For details, see [Double-Precision Executables in the CFX-Solver Manager User's Guide](#).

4. If required, under **Interpolator Memory**, adjust the memory configuration. For details, see [Configuring Memory for the CFX-Solver in the CFX-Solver Manager User's Guide](#).

2.2.8. Configuring Memory for the CFX-Solver

There may be instances when the CFX-Solver fails due to insufficient 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 \(p. 123\)](#).

2.3. 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 \(p. 36\)](#).

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 \(p. 73\)](#).

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

The following dialog boxes may appear:

- **Solver Run Finished Normally**
- **Solver Run Stopped By User**
- **Solver Run Terminated With Errors**

To close the dialog box without post-processing the results, click **OK**.

On the **Solver Run Finished Normally** and **Solver Run Stopped By User** dialog boxes, you can choose from the following options:

Post-Process Results

Opens CFD-Post with the specified CFX-Solver Results file loaded.

Multi-Configuration Load Options

If this option is available, you can use it to control how the results of a multi-configuration run are loaded, or to load just the last case of such a run.

Select a multi-configuration load option to control how you load a multi-configuration (.mres) file or a results file (.res) that contains a run history (that is, a file that was produced from a definition file that had its initial values specified from a results file from a previous run and saved to the results file that you are loading). Choose:

- **Single Case** to load all configurations of a multi-configuration run as a single case, or all of the results history from a results file that contains a run history. In either case, only one set of results will appear in the viewer, but you can use the timestep selector to move between results. This option is not fully supported.
- **Separate Cases** to load all configurations from a multi-configuration run into separate cases. If a results file with run history is loaded, CFD-Post loads the results from this file and the results for any results file in its run history as separate cases. Each result appears as a separate entry in the tree.
- **Last Case** to load only the last configuration of a multi-configuration results file, or only the last results from a results file that contains a run history.

Shut down CFX-Solver Manager

Shuts down CFX-Solver Manager before CFD-Post is launched. This minimizes the number of licenses you use concurrently.

To post-process the results, make the desired settings on the dialog box and click **OK**.

After a run has finished you can use CFX-Solver Manager to:

- Define a new run by following the earlier procedure. For details, see [Define Run Command \(p. 93\)](#)
- Calculate more timesteps for the original run. For details, see [Restarting a Run \(p. 21\)](#).
- 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 in the CFD-Post User's Guide](#).
- Print residual plots in the convergence history plots. For details, see [Printing an Image of the Convergence History \(p. 3\)](#).
- 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 \(p. 123\)](#).
- Export results in a format suitable for post-processors other than CFD-Post. For details, see [File Export Utility \(p. 149\)](#).

- 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 [CFX-Solver Manager File Menu](#) (p. 93).

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 tutorials.

2.4. 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](#) (p. 19)
- [General Procedure](#) (p. 19)
- [Configuring a Parallel Run](#) (p. 19)

2.4.1. Overview

There are several parallel run modes, including PVM (Parallel Virtual Machine) and MPI (Message Passing Interface) communication libraries. Both are simply libraries that allow the flow solver processes to communicate with each other. An open-source version of MPI, MPICH, implemented at Argonne National Labs, enables communication over standard TCP/IP networks, as well as through shared memory. Proprietary, vendor specific versions of MPI are available on 64-bit Linux 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 in [Using the Solver in Parallel in the CFX-Solver Modeling Guide](#).

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

2.4.2. 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.

2.4.3. Configuring a Parallel Run

1. Follow the procedure for a serial run.

For details, see [Solver Run Overview](#) (p. 9).

2. Under **Run Mode**, select a parallel method:

- [Local Parallel Setup](#) (p. 20)
- [Distributed Parallel Setup](#) (p. 20)

2.4.3.1. 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 \(p. 13\)](#). The solver enables further changes. For details, see [Solver Tab \(p. 16\)](#).

1. Select one of the local parallel run modes (for example, Platform 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 \(p. 13\)](#).

2.4.3.2. Distributed Parallel Setup

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

2.4.3.2.1. 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 14.0, it operated on the `hosts.ccl` file. This change was made necessary by a CCL change.

2.4.3.2.2. 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.

2.4.3.2.3. 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 \(p. 13\)](#).

2.5. Restarting a Run

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

- The CFX-Solver stopped prematurely, at your 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 \(p. 21\)](#)
- [Restart Details \(p. 21\)](#)

2.5.1. 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 [CFX-Solver Manager File Menu \(p. 93\)](#).

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

2.5.2. 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 \(p. 22\)](#)
- [Runs After Physical Model or Solver Parameter Changes \(p. 22\)](#)
- [Runs After Topology or Mesh Changes \(p. 22\)](#)

- [Multi-Configuration Simulations](#) (p. 23)

2.5.2.1. Runs Using Mesh Adaption

Restarting a run that 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.

2.5.2.2. 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 as well as 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 that 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 in the CFX-Solver Modeling Guide](#).

Additional information on initial conditions is available in the [Initialization in the 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 in the CFX-Solver Modeling Guide](#).

2.5.2.3. 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
- Changing the name or location of any boundary condition, including the default boundary condition

...then you can continue the run as described in [Runs After Physical Model or Solver Parameter Changes](#) (p. 22). 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.

2.5.2.4. 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 in the 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 enables 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.

2.6. ANSYS Multi-field Run

A tutorial is provided to demonstrate an ANSYS Multi-field run. For details, see ["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](#) (p. 24)
- [General Procedure](#) (p. 24)
- [Monitoring a Run in Progress](#) (p. 26)
- [Monitoring a Completed Run](#) (p. 27)
- [ANSYS Multi-field Residual Plotting](#) (p. 27)
- [Processing the ANSYS Input File](#) (p. 27)
- [Restart Procedure for ANSYS Multi-field Runs](#) (p. 29)

2.6.1. Overview

An ANSYS Multi-field run enables 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 in the CFX Reference Guide](#) and [Coupling CFX to an External Solver: ANSYS Multi-field Simulations in the CFX-Solver Modeling Guide](#).

2.6.2. General Procedure

The steps that take place when a full ANSYS Multi-field run (where **MF 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 selected, 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. 27).
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:

- [Starting ANSYS CFX and ANSYS Separately](#) (p. 25)
- [Process ANSYS Input File Only](#) (p. 25)
- [Directory Structure](#) (p. 26)
- [Starting an ANSYS Multi-field Run from the Command Line](#) (p. 26)

If a run is to be restarted, a different procedure applies. This is described in [Restart Procedure for ANSYS Multi-field Runs](#) (p. 29).

2.6.2.1. 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 that tell the CFX-Solver how to communicate with the ANSYS solver that 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 CFX 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`.

2.6.2.2. Process ANSYS Input File Only

The processing of an ANSYS input file is a procedure that 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 that 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. 29). 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 clear **Process ANSYS Input File**.

The processing step is described more fully in [Processing the ANSYS Input File \(p. 27\)](#).

2.6.2.3. 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.

While 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.res` 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 \(p. 26\)](#).

2.6.2.4. 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. For example:

```
cfx5solve -def file.def -double -mfx-run-mode "Start\ ANSYS\ and\ CFX" \  
-ansys-input file.inp -ansys-arguments "-np\ 2"
```

This starts both solvers using the files specified, running CFX in double-precision and using two partitions for ANSYS. Note that the escape character "\" is used before spaces in the arguments as shown and is necessary on some systems.

2.6.3. 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 \(p. 26\)](#). 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-Solver 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**.

2.6.4. Monitoring a Completed Run

A completed ANSYS Multi-field run can only be monitored in the CFX-Solver Manager if either a CFX-Solver Results file or a CFX-Solver Output file exists for the run. If either of these files does exist, then a completed run can be monitored in the same way as an ordinary CFX-Solver 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-Solver 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**.

2.6.5. 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. 80) and *Plot Lines Tab* (p. 103).

2.6.6. Processing the ANSYS Input File

The processing of an ANSYS input file is a procedure that 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 that 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.

The following topics are discussed in this section:

- *When the ANSYS Input File Needs Processing* (p. 27)
- *The Processing Step* (p. 28)

2.6.6.1. 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.

2.6.6.2. 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, `MFDI, 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 in the 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 that 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 in the CFD-Post User's Guide](#).

2.6.7. Restart Procedure for ANSYS Multi-field Runs

Restarting an ANSYS Multi-field run involves restarting both the ANSYS and CFX Solvers, in such a 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](#) (p. 29)
- [Restart from Intermediate Point During Previous Run](#) (p. 29)
- [Changing CFX Physics or ANSYS Multi-field Settings](#) (p. 31)
- [Changing ANSYS Physics](#) (p. 31)
- [Restart Limitations](#) (p. 32)

2.6.7.1. Restart from End of Previous Run

This is the most straightforward 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 setup 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 rewriting 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.

The **Continue History From** check box should always be selected for Multi-field restarts, otherwise the Mesh Displacements will be reset to zero. See [Continuing the History in the CFX-Solver Modeling Guide](#) for further details.

3. On the **MultiField** tab, enable the **Restart ANSYS Run** toggle, and browse to find the database (*.db) 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.

2.6.7.2. Restart from Intermediate Point During Previous Run

You may want to restart from an intermediate point of a previous run if the run failed to complete successfully. Restarting from an intermediate point is possible only if you requested restart files to be written by both the CFX and ANSYS solvers at the same coupling step in the run history. Restart files

for the CFX solver can be created as usual, by writing backup files at an appropriate frequency. Restart files for the ANSYS solver can be created using the MFOU and MFRC commands, for example:

```
MFOU,10
MFRC,50,1
```

The above commands create ANSYS results data every 10th coupling step (MFOU,10) and restart data every 50th coupling step, with only the most recent restart data kept (MFRC,50,1). See the ANSYS documentation for further details on these commands. When restart data is written, results data must also be written for a successful restart (the MFRC frequency must be a multiple of the MFOU frequency). The above commands can be added to the ANSYS input file directly or to a Commands object in the relevant Solution item in the tree within Simulation. The ANSYS run directory will contain a *.rdb file when restart data is written and this should be used for the restart. Assuming that the restart from an intermediate point is available, the procedure is as follows:

1. **Coupling Initial Time** should be left as `Automatic`. This will pick up the most recent restart time from the *.rdb file. If you want to restart from an earlier time point, the Coupling Initial Time should 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 rewriting 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 backup file or the full transient results file corresponding to the restart point as the CFX-Solver input file, or as the Initial Values file if a Solver input file has already been provided. The backup or full transient file should be consistent with the restart time used in the ANSYS *.rdb file.

The **Continue History From** check box should always be enabled for Multi-field restarts, otherwise the Mesh Displacements will be reset to zero. See [Continuing the History in the CFX-Solver Modeling Guide](#) for further details.

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. Note that when starting through the CFX-Solver Manager, the Mechanical Input File must still be specified but it is not used. When starting from the CFX-Solver script, the Mechanical Input File does not need to be provided.

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

- Restarting from an intermediate point is limited to ANSYS runs that 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 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.

Performing a restart from an intermediate point of a run that was itself a restart has additional limitations:

- When a run is restarted from the end of the previous run, the *.db file is copied from the initial Mechanical run to the new Mechanical working directory (assuming that you selected the *.db file from the initial Mechanical run, as is recommended). However, the *.rdb file is not copied as it is not used. Even if this second run then writes out restart files, the *.rdb file is not written out again by the Mechanical solver into the new working directory. If it is later desired to restart from one of these restart

files from the second run, then the `*.rdb` file from the initial run must be manually copied from the initial run into the same directory as the later Mechanical restart files, before the subsequent restart can be performed.

- A run that is restarted from an `*.rdb` file will run using the settings from the initial run. Any subsequent changes to the Mechanical problem definition, such as at the start of the first restarted run, will be disregarded unless they are reapplied.
- If any run is a steady-state run, then subsequent runs may have problems with synchronization of the output of Mechanical restart files and the output of CFX full transient/backup files. For example, if the first run is a steady-state run, then the typical setup has ANSYS performing one load step and CFX performing multiple iterations. Therefore the total step count of CFX and ANSYS will not be the same, and files output using the MFRC command every N ANSYS steps will not necessarily be synchronized with the CFX transient/backup files output every N CFX timesteps/iterations. There is no easy way to avoid this other than tightly controlling how many iterations are done by each solver to avoid the synchronization issues. For example, one way to set this up would be to have the initial steady-state case calculate N load steps (where N is the desired frequency of output of the restart files) and to either carefully limit the number of CFX iterations, or to perform the restart for CFX with **Continue History** off so that the CFX iteration count is started from zero again.

2.6.7.3. 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.

2.6.7.4. 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 **MF 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 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 **MF 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** check box is still selected, 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.

2.6.7.5. Restart Limitations

Restarted simulations will give results that are identical to continuous simulations only if fields and loads are converged within each multi-field/coupling timestep and ANSYS solves first within each stagger/coupling iteration.

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.

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 (that is, in serial or parallel).

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

[3.1. Files Used by the CFX-Solver](#)

[3.2. Files Generated by the CFX-Solver](#)

3.1. 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 (that is, 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 (that is, no run history) starting from iteration or time step # 1.

Single Configuration Simulations

Solver Input File	Initial Values Specification	Description
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 the Initial Values object(s).

Multi-Configuration Simulations

Solver Input File	Configuration-Specific Initial Values Specification	Description
CFX-Solver Input file (.mdef)		New simulation (that is, 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 (for example, via the <code>cfx5stop</code> command). Note that this 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 (that is, 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 in the 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 in the CFX-Solver Modeling Guide](#) for details.
- Unsetting (that is, 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 in the 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 in the CFX-Solver Modeling Guide](#) for details.

3.2. 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 following sections:

- [CFX-Solver Output File](#) (p. 36)
- [CFX-Solver Output File \(Transient Runs\)](#) (p. 51)
- [CFX-Solver Output File \(Interpolation Runs\)](#) (p. 54)
- [CFX-Solver Output File \(Parallel Runs\)](#) (p. 55)
- [CFX-Solver Output File \(Mesh Adaption Runs\)](#) (p. 57)
- [CFX-Solver Output File \(Remeshing Runs\)](#) (p. 58)
- [CFX-Solver Output File \(Conjugate Heat Transfer Runs\)](#) (p. 58)
- [CFX-Solver Output File \(GGI Runs\)](#) (p. 59)
- [CFX-Solver Output File \(Combustion Runs\)](#) (p. 60)
- [CFX-Solver Output File \(Particle Runs\)](#) (p. 62)
- [CFX-Solver Output File \(ANSYS Multi-field Runs\)](#) (p. 66)
- [CFX-Solver Output File \(Radiation Runs\)](#) (p. 67)
- [CFX-Solver Output File \(Rigid Body Runs\)](#) (p. 69)
- [CFX-Solver Results File](#) (p. 69)

Additional files are generated for the run as follows:


```

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
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: computer123
    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

```

3.2.1.3. 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

```

3.2.1.4. 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

3.2.1.5. 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
- The area fractions of mesh interfaces that were unmapped.

Mesh quality diagnostics include measures of mesh orthogonality, expansion and aspect ratio (see [Mesh Issues](#)). For each measure, there are value ranges that are considered good, acceptable, and poor (that is, 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 (that is, 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
------------------------	---	------

3.2.1.8. 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
```

3.2.1.9. Convergence History

The convergence history section details the state of the solution as it progresses. Equation residual information at specified locations enables 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 \(p. 43\)](#) 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.

$$\text{Rate} = \frac{R_n}{R_{n-1}} \quad (3-1)$$

- 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 CFX-Solver Output 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 |
+-----+-----+-----+-----+
```

3.2.1.10. 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
```

3.2.1.11. 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>
```

3.2.1.12. 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 "MaximumOver 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 in the 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
-----

```

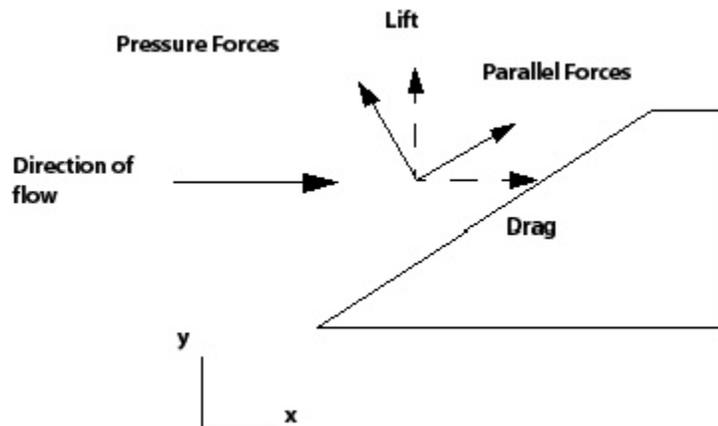
```

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 %

```

3.2.1.13. 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 because 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 vector F_t and the position vector r . As an example, review [Equation 3-2 \(p. 48\)](#) where M_n and M_t are the pressure and viscous moments respectively.

$$\begin{aligned} M_n &= \vec{r} \times \vec{F}_n \\ M_t &= \vec{r} \times \vec{F}_t \end{aligned} \tag{3-2}$$

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

3.2.1.14. 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

3.2.1.15. 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          |
+-----+-----+-----+
```

3.2.1.16. 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
```

3.2.1.17. 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   |
+-----+-----+-----+
```

3.2.1.18. 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	

3.2.1.19. 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.

```

3.2.2. 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

3.2.2.1. 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 |
+-----+-----+-----+-----+

```

3.2.3. CFX-Solver Output File (Transient Blade Row Runs)

The following sections describe some output that occurs in transient blade row simulations:

[3.2.3.1. Post-processing Information](#)

[3.2.3.2. Fourier Transformation Information](#)

[3.2.3.3. Stability Information \(Time Transformation Runs\)](#)

3.2.3.1. Post-processing Information

There is a table that indicates, for each domain, the starting and ending time steps for the data compression algorithm, as well as the fundamental period.

An example of this table follows:

```

=====
| Transient Blade Row Post-processing Information |
+-----+-----+-----+-----+
| | | | Fourier coefficient |

```

Domain Name:	Disturbance period	accumulation time step range	
		Start	End
R1	4.7619E-04	6120	6179
S2	4.0816E-04	6120	6179

A description of the items in this table follows:

- Domain Name: The name of a domain that accumulates transient blade row data.
- Disturbance period: The fundamental periods (and their harmonics) used for the data compression algorithm. The period is given in time units as specified by the solution units. For details on solution units, see [Setting the Solution Units in the CFX-Pre User's Guide](#).
- Start: The time step at which the data compression algorithm starts for the applicable domain.
- End : The time step at which the data compression algorithm ends for the applicable domain.

Note

If the solver run is stopped before the data compression algorithm finishes for all the listed domains, no data will be visible in the post-processor.

3.2.3.2. Fourier Transformation Information

For Fourier transformation cases, there is a table that lists information about time steps and coefficient loops.

An example of this table follows:

```

=====
|                               Fourier Transformation Information                               |
=====
|                                                                                               |
+-----+-----+-----+-----+
|                               | Max.   | Time Step Range | |
|                               | Coeff. |                   |
|                               | Loops  | Start           | End           |
+-----+-----+-----+-----+
| Initialisation Period      | 3      | 41              | 160          |
+-----+-----+-----+-----+
| Startup Period             | 3      | 161             | 220          |
+-----+-----+-----+-----+
| Full Fourier Transformation | 10     | 221             | 640          |
=====

```

3.2.3.3. Stability Information (Time Transformation Runs)

For time transformation cases, there is a table that lists the acceptable range of pitch ratio for a stable run.

This pitch ratio is computed as follows:

- For inlet disturbance cases with a stationary passage and a moving signal, the pitch ratio is computed as the passage pitch divided by the signal pitch.
- For inlet disturbance cases with a moving passage and a stationary signal, the pitch ratio is computed as the signal pitch divided by the passage pitch.

- For rotor-stator cases, the pitch ratio is computed as the pitch of the stator divided by the pitch of the rotor.

An example of this table follows:

```

=====
|                               Time Transformation stability limits                               |
| Pitch ratio computed using stationary/rotating component pitches                             |
+-----+-----+-----+-----+
|                               Pitch ratio                               |
| Disturbance name                |-----|-----|-----|
|                               | Minimum | Maximum | Current |
+-----+-----+-----+-----+
| Time Transformation 1           | 0.60   | 1.40   | 1.33 Poor |
+-----+-----+-----+-----+
| Poor: The pitch ratio is close to one of the stability limits. |
| If the solution becomes unstable please do one of the         |
| following:                                                       |
| (1) Modify the number of passages per components to obtain a  |
| pitch ratio within the stability limits.                         |
| (2) For further information on Time Transformation stability   |
| limits please consult the Transient Blade Row guide.          |
+-----+-----+-----+-----+

```

3.2.4. 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

3.2.5. 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.

3.2.5.1. 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
```

3.2.5.2. 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 (PVM)

Host computer:     Speedy (PID:4296)
Par. Process:      Master running on mesh partition:      1
Solver Build:      Thu Feb  4 09:22:42 CET 2010 by user12
Attributes:        single-int32-64bit-novc8-noifort-novc6-...
Job started:       Thu Feb  4 09:23:33 2010

Host computer:     Speedier (PID:4297)
Par. Process:      Slave running on mesh partition:       2
Solver Build:      Thu Feb  4 09:22:42 CET 2010 by user12
Attributes:        single-int32-64bit-novc8-noifort-novc6-...
Job started:       Thu Feb  4 09:23:33 2010
```

Note

Slave processes continue to run after the master process has been terminated. The OUT file's Job Information area lists the process IDs and hostname used in the parallel processing so that you can manually end those processes using the appropriate system commands.

3.2.5.3. 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 %    |
|-----+-----+-----+-----+-----+
```

3.2.5.4. 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

3.2.5.5. 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.

```

3.2.6. 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

```

3.2.7. 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\ballvalvel_001\5_oldmesh.res

  Text output generated during remeshing has been written to:
    d:\ICEMRemesh\ballvalvel_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.

3.2.8. CFX-Solver Output File (Conjugate Heat Transfer Runs)

3.2.8.1. Thermal Energy Flow Through a 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 %

```

3.2.8.2. Thermal Energy Flow Between the Fluid and Solid within a Porous Domain

When a solid in a porous domain is included, CFX-Solver Manager reports the thermal energy that flows between the fluid and the solid (`Domain Src (Neg)` in the sample below), and from the solid into and out of existing boundaries (`Boundary` in the sample below). If the solid has a source term, the total thermal energy (`Sub-Domain` in the sample below) that is added is included in the output file, along with the global imbalance.

The section for thermal energy for a solid in a porous domain looks similar to the following:

```

+-----+
|          Boundary T-Energy - 2 Flow and Total Source Term Summary          |
+-----+
Boundary       : Default Solid                               -4.9569E+00
Boundary       : Default Fluid Solid2                      -3.4767E+00
Domain Src (Neg) : Porous                                  -1.3999E+00
Domain Src (Pos) : Porous                                   5.8335E+00
Sub-Domain     : Porous Subdomain 1                        4.0001E+00
-----
Domain Imbalance :                                          5.1498E-05
Domain Imbalance, in %:                                    0.0009 %

```

3.2.9. 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, and so on, 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 |
+-----+-----+-----+-----+-----+

```

3.2.10. 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.

3.2.10.1. Multicomponent Specific Enthalpy Diffusion (MCF)

This applies also to non-reacting multicomponent flow. For multicomponent fluids with heat transfer, the diffusion term assembly modes are reported for the molecular and the turbulent diffusion, respect-

ively. The option for each part may be either generic assembly or unity Lewis number assumption, $Le = 1$

```
+-----+
|           Multi-Component Specific Enthalpy Diffusion           |
+-----+
```

Enthalpy equation assembly for secondary terms derived from multi-component species transport. Molecular and turbulent transport may use either Unity Lewis Number ($Le=1$) assumption or generic assembly.

```
Domain Name : Default Domain
Mixture                                           (Le_mol=gen, Le_trb=1 )
```

3.2.10.2. 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
```

3.2.10.3. 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
```

3.2.10.4. 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/Ultime 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

```

3.2.11. CFX-Solver Output File (Particle Runs)

3.2.11.1. 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. 43).

3.2.11.2. 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 type | Fate type | Particles |
+-----+
| Sand Fully Coupled | Entered domain | : | 200 |
| | Left domain | : | 200 |
+-----+

```

The following fate types may be encountered during particle tracking:

3.2.11.2.1. 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 in the CFX-Pre User's Guide](#).

3.2.11.2.2. Continue from Last Time Step (Transient Only)

See [Waiting for Next Time Step \(Transient only\)](#) (p. 64).

3.2.11.2.3. 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 in the CFX-Solver Modeling Guide](#).

3.2.11.2.4. Entered Domain

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

3.2.11.2.5. 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](#). Any remaining mass,

momentum, and energy that must be exchanged with the fluid to equilibrate the aborted particle with the local fluid properties, is exchanged in the current control volume.

3.2.11.2.6. 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](#). Any remaining mass, momentum, and energy that must be exchanged with the fluid to equilibrate the aborted particle with the local fluid properties, is exchanged in the current control volume.

3.2.11.2.7. 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](#). Any remaining mass, momentum, and energy that must be exchanged with the fluid to equilibrate the aborted particle with the local fluid properties, is exchanged in the current control volume.

3.2.11.2.8. 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](#).

3.2.11.2.9. 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.

3.2.11.2.10. 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.

3.2.11.2.11. 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 in the CFX-Pre User's Guide](#).

3.2.11.2.12. 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.

3.2.11.3. 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 in the 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

```

3.2.11.4. 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](#) and [Particle Under Relaxation Factors](#) in the [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 in the CFX-Solver Modeling Guide](#).

3.2.11.5. 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 MON-


```

+-----+
| 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 |
-----

```

3.2.13. CFX-Solver Output File (Radiation Runs)

3.2.13.1. Convergence History

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

3.2.13.1.1. 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 |
+-----+-----+-----+-----+-----+

```

3.2.13.1.2. 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.

3.2.13.1.3. 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`). The presence of values greater than 30% indicates that the value of **Number of Histories** is too small to resolve the radiation field. The presence of small isolated boundary regions with values of `Irradiation Flux.Normalized Std Deviation` larger than 30% is an indication that the element faces in those regions were insufficiently sampled.
- `%SD Vol` is the maximum normalized standard deviation of the radiation intensity within a finite element (`Radiation Intensity.Normalized Std Deviation`). Similar to `%SD Sur`, the values of `Radiation Intensity.Normalized Std Deviation` are expected to be less than 30%.
- `%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.

3.2.13.2. Summary Fluxes

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

$$q_{\text{spec}} = q_{\text{rad}} + q_{\text{cond}} \quad (3-3)$$

- 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.

$$q_{\text{rad}} + q_{\text{cond}} \quad (3-4)$$

- The CPU Requirements of Numerical Solution table reports the amount of CPU time spent due to the use of a thermal radiation model.

3.2.14. CFX-Solver Output File (Rigid Body Runs)

For runs involving Rigid Body modeling, the CFX-Solver outputs convergence information at regular intervals according to the update frequency specified in the **Solver Control** details view on the **Rigid Body Control** tab in CFX-Pre.

```

-----
COEFFICIENT LOOP ITERATION =      5                CPU SECONDS = 1.461E+01
-----
|                               Rigid Body Convergence                               |
+-----+-----+-----+-----+
|                               | Quantity          Change          Rate          |
+-----+-----+-----+-----+
| rigidBody1                    | Motion          1.783E-01    0.26          |
|                               | Force           7.343E-01    1.04          |
+-----+-----+-----+-----+

```

3.2.15. 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 in the CFX-Pre User's Guide](#) and [Transient Results Tab in the CFX-Pre User's Guide](#). For details on how to create backup results files while the run is executing, see [Backup Run Command \(p. 108\)](#). 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`.

3.2.15.1. 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 that are suitable for use with other post-processors by using the CFX Export facility. For details, see [File Export Utility \(p. 149\)](#).

3.2.15.2. 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 \(p. 33\)](#).

3.2.16. 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

```

3.2.16.1. 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 that 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.

3.2.16.2. 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-gray models the emission and scattering coefficients are the spectrum integrated coefficients. In a gray model the absorption coefficient equals the emission coefficient, in a non-gray 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-gray models will also have the band by band cooling.

3.2.16.3. 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-gray model. Grey surfaces have an albedo equal to one minus the emissivity, in the non-gray 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-gray 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 because 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.

3.2.17. 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.

3.2.18. 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 in the 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. 27).

3.2.19. 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`.

3.2.19.1. Header

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

3.2.19.2. 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, and so on) and simulation control specifications (such as configuration definitions).


```

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 = Platform 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
  END

```

```

MEMORY CONTROL:
  Memory Allocation Factor = 2
END
PARALLEL ENVIRONMENT:
  Start Method = Platform MPI Local Parallel
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 = mcfgl.mdef
END
END
END

```

3.2.19.3. 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
- You stop 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 in the CFX-Pre User's Guide](#) and [Termination Control in the 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 (for example, whether it succeeded or failed, or whether you stopped the run while it was 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.


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

* User: Control Condition 1
```

3.2.20. 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 \(p. 69\)](#)) 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:

- 4.1. Equation Residual
- 4.2. Convergence Results and RMS
- 4.3. Transient Residual Plotting
- 4.4. Residual Plotting for ANSYS Multi-field Runs

4.1. 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 in the 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 in the 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.

4.2. 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 $5.0\text{E-}4$. 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 \(p. 100\)](#).

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 in the 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.

4.3. 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 within each timestep. Each timestep is divided by the number of inner coefficient loops. The values are produced for all variables in each coefficient loop.

With the plotting selected, CFX-Solver Manager outputs the graph of monitor data. The strategy that CFX-Solver Manager uses for plot data is to plot the loops evenly across the space between iteration N-1 and N, so that the last coefficient loop is in line with iteration N on the plot. When the first coefficient loop value becomes available, it is the *only* value, and therefore it is placed in line with iteration N. As more data is added, the older coefficient loop values are shifted back, so that they are evenly spaced, with the last coefficient loop always in line with N.

Example: Assume that four coefficient loops were run for timestep 39. The first coefficient loop 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.

For details on plotting coefficient loop data, see [Global Plot Settings Tab \(p. 106\)](#).

4.4. 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 in the CFX-Solver Modeling Guide](#)
- [ANSYS Field Solver Plots](#) (p. 81)
- [ANSYS Interface Loads Plots](#) (p. 81)

For the extra plots to be displayed, the CFX-Solver Manager must have started CFX-Solver by selecting an **MF Run Mode** of either `Start ANSYS and CFX` or `Start ANSYS only` (see [MultiField Tab](#) (p. 12)), 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.

4.4.1. 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 [Newton-Raphson Procedure in the Mechanical APDL Theory Reference](#).

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.

4.4.2. 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:

$$\Phi = \frac{\sqrt{\sum (u_{new} - u_{old})^2}}{\sqrt{\sum u_{new}^2}} \quad (4-1)$$

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 $\Phi < \Phi_{min}$, where Φ_{min} is the convergence target for that quantity

set in CFX-Pre (see [Solver Controls, External Coupling Tab in the 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 \(p. 81\)](#) 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:

$$e = \frac{\log(\Phi/\Phi_{\min})}{\log(10/\Phi_{\min})} \quad (4-2)$$

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

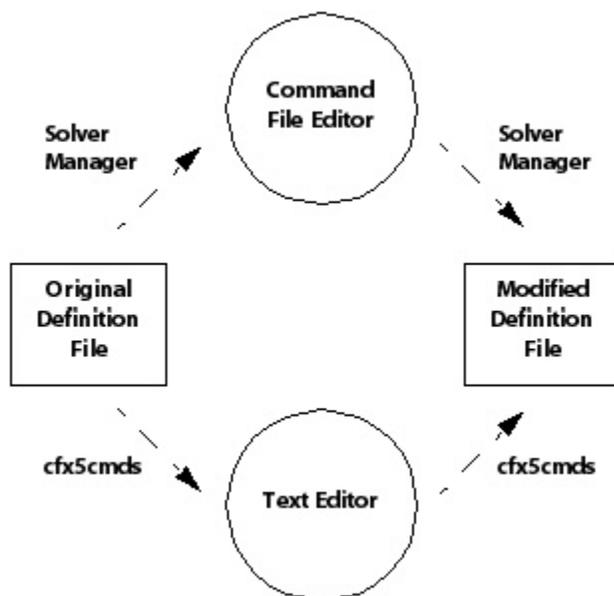
This chapter discusses the details of editing existing CFX-Solver input (or results) files to make minor changes to a CFD model, without having to use CFX-Pre:

- 5.1. Workflow Overview
- 5.2. Command File Editor Overview
- 5.3. Menus in the Command File Editor
- 5.4. Command File Editor Rules
- 5.5. Command File Editor Appearance
- 5.6. Editing the Command Language (CCL) File
- 5.7. Command Language File Rules
- 5.8. RULES and VARIABLES Files

5.1. 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](#) (p. 83).

Figure 5.1 CFX-Solver Input File Modification Workflows



5.2. 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 enables modification of parameter settings and the addition of new parameters to existing CFX-Solver input files as required.

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

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

See *Edit CFX-Solver File Command* (p. 113).

- **Tools > Edit Run In Progress**

See *Edit Run In Progress Command* (p. 118).

- **Tools > Edit Current Results File**

See *Edit Current Results File Command* (p. 119).

5.3. Menus in the Command File Editor

The Command File editor has three menus:

- *File Menu* (p. 84)
- *Edit Menu* (p. 84)
- *Help Menu* (p. 87)

5.3.1. File Menu

The **File** menu contains:

- *Save Command* (p. 84)
- *Validate Command* (p. 84)
- *Exit Command* (p. 84)

5.3.1.1. Save Command

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

5.3.1.2. 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.

5.3.1.3. Exit Command

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

5.3.2. Edit Menu

The **Edit** menu contains:

- *Add Parameter Command* (p. 85)
- *Edit Parameter Command* (p. 85)
- *Delete Parameter Command* (p. 86)
- *Add Expert Parameter Section Command* (p. 86)
- *Find Command* (p. 87)

- [Find Next Command](#) (p. 87)

5.3.2.1. Add Parameter Command

Some categories (for example, 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. 86).

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 enables 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. 87).

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.

5.3.2.2. Edit Parameter Command

The **Edit Parameter** option enables 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.

5.3.2.2.1. 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.

5.3.2.2.2. 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. 87).

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.

5.3.2.3. Delete Parameter Command

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

5.3.2.4. 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. 85).

Once the value for an expert parameter has been set, it can be edited as needed. For details, see [Edit Parameter Command](#) (p. 85).

5.3.2.5. Find Command

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

5.3.2.6. Find Next Command

Finds the next occurrence of the keyword or keywords.

5.3.3. Help Menu

The **Help** menu contains:

- Help [On CFX Solver File Editor Command](#) (p. 87)
- Help [On Expert Parameters Command](#) (p. 87)

5.3.3.1. On CFX Solver File Editor Command

The help associated with the Command File editor is launched.

5.3.3.2. On Expert Parameters Command

The CFX-Solver Expert Control Parameters help is launched.

5.4. Command File Editor Rules

If you are unsure about which values are appropriate 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. 88).

5.5. 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 the line. This opens the line in a configurable dialog box for viewing and editing.

5.6. Editing the Command Language (CCL) File

In some circumstances, more significant changes may be required to the CFX-Solver input file than is possible through the Command File editor. ANSYS 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 that is set up correctly to run CFX commands. For details, see [Command Line in the CFX Reference Guide](#).

5.7. 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 [Setting the Solution Units in the CFX-Pre User's Guide](#).
- Everything apart from expert parameter names is case-sensitive. Use care to distinguish upper case letters from lowercase letters. You should use lowercase letters for expert parameter names to match those shown in the Command File Editor Help's **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 declarations 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 \(p. 89\)](#).

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
  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.

5.8. 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.

5.8.1. 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 it 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, and so on.

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

5.8.2. RULES File

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

5.8.2.1. 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.

5.8.2.2. 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
```

5.8.2.3. 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.

5.8.2.4. 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 that 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. 118).

Chapter 6: CFX-Solver Manager 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 \(p. 10\)](#).

Monitor Run in Progress Command

Used when CFX-Solver Manager has been closed and a run currently underway in the Solver must 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 Workspace 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 \(p. 93\)](#) or [Monitor Finished Run Command \(p. 93\)](#), as appropriate.

Quit Command

To quit CFX-Solver Manager, select **File > Quit**. 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 \(p. 93\)](#).

Chapter 7: CFX-Solver Manager Edit Menu

The **Edit > Options** dialog box 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. For descriptions of the available options, see:
 - [CFX-Solver Manager Options](#) (p. 95)
 - [Common Options](#) (p. 96)

If desired, you can use the **CFX Defaults** or the **Workbench Defaults** buttons at the bottom of the dialog box to quickly set CFX-Pre, CFX-Solver Manager, and CFD-Post to have the standard operation of CFX or Workbench respectively. The only settings visible in CFX-Solver Manager that are affected by these buttons are **Common > Viewer Setup > Mouse Mapping**.

3. Click **OK**.

7.1. 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. 118).

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.

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 \(p. 107\)](#).

7.2. 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.

7.2.1. Appearance

The appearance of the user interface can be controlled from the **Appearance** options. The default user interface style will be set to that of your machine. For example, on Windows, the user interface has a Windows look to it. If, for example, a Motif appearance to the user interface 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.

7.2.2. Viewer Setup

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

For details, see [Double Buffering](#) (p. 97).

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

For details, see [Unlimited Zoom](#) (p. 97).

7.2.2.1. 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.

7.2.2.2. Unlimited Zoom

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

7.2.2.3. 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 stand-alone mode. To adjust or view the mouse mapping options, select **Edit > Options**, then **Viewer Setup > Mouse Mapping**. For details, see [Mouse Button Mapping](#).

7.2.3. 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 in the main **Options** dialog box; 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 in this dialog box.

For example, if you have set **Velocity** to $[m \ s^{-1}]$ in this dialog box 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: CFX-Solver Manager Workspace Menu

The **Workspace** menu controls layout, plots, and text windows that 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 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 the commands available from the **Workspace** menu:

- 8.1. Workspace Properties Command
- 8.2. New Monitor Command
- 8.3. Stop Current Run Command
- 8.4. Restart Current Run Command
- 8.5. Backup Run Command
- 8.6. Arrange Workspace Command
- 8.7. Toggle Layout Type Command
- 8.8. Load Layout Command
- 8.9. Save Layout Command
- 8.10. View RMS Residuals Command
- 8.11. View MAX Residuals Command
- 8.12. Reset to Default Workspace Command
- 8.13. Close Workspace Command

8.1. Workspace Properties Command

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

By default, appropriate monitors are automatically created by CFX-Solver Manager for the particular type of simulation you are running. However, the type and/or number of plots that appear can be changed by using **Workspace > Workspace Properties**. This launches the **Workspace Properties** dialog box, which has the following tabs:

- *General Settings Tab* (p. 99)
- *Monitors Tab* (p. 100)
- *Global Plot Settings Tab* (p. 106)

8.1.1. General Settings Tab

The **General Settings** tab displays the following workspace information, which cannot be edited.

- **Out File** displays the path and name of the CFX-Solver 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. 93) and *Run Definition Tab* (p. 11). Information on selecting different runs is available in *Workspace Selector* (p. 2).

8.1.2. Monitors Tab

Not all of the monitors relevant to a solution are displayed by default; however, you can add them in the **Monitors** tab. The following types of monitors are available:

- *Plot Monitor* (p. 100)
- *Residual Monitor* (p. 100)
- *Text Monitor* (p. 100)

8.1.2.1. Plot Monitor

Plot monitors show the values of expressions versus timestep.

8.1.2.2. Residual Monitor

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

8.1.2.3. Text Monitor

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

8.1.2.4. 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. The **All** setting displays the full list of available monitors.

8.1.3. 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. 101)
 - [Range Settings Tab](#) (p. 102)
 - [Plot Lines Tab](#) (p. 103)

8.1.4. 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. 101)
 - [Range Settings Tab](#) (p. 102)
 - [Plot Lines Tab](#) (p. 103)

Note

Alternatively, to access the **Monitor Properties** dialog box, right-click the graph area of any of the monitor tabs.

8.1.4.1. Deleting Monitors

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

8.1.4.2. 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. 101)
- [Range Settings Tab](#) (p. 102)
- [Plot Lines Tab](#) (p. 103)

8.1.4.2.1. General Settings Tab

1. Under **Window Label**, type the name to display for the monitor.
2. If working with a text monitor:

- a. Under **Text File Name**, click *Browse*  and select a file containing the definition for the text monitor.

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

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

If selected, the legend is displayed in the monitor.
 - c. 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.
 - d. Under **Grid Color**, click *Color Selector*  and set the grid color.
4. Select or clear **Visibility**.

If selected, the monitor is displayed.

8.1.4.2.2. Range Settings Tab

1. Under **Timestep Range Mode**, select from the following options:
 - *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 that 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.
 - *This Run and Previous Timestep* displays all timesteps for the current run and the last timestep from any previous run. By selecting this option, plots will contain all data generated in the current run and will also display the coefficient loop data for the previous timestep from any previous run from which the current run was started.
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.

Note

If you accidentally specify a lower bound that is higher than the upper bound, CFX-Solver Manager will reverse the values internally to determine the range, but the values you entered will not be changed in the user interface.

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**.

8.1.4.2.3. Plot Lines Tab

Variables that 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 enables the specification of all variables relating to the CFX run.
- **ANSYS Field Solver:** This option is available only if an ANSYS Multi-field run is being performed with ANSYS running a nonlinear simulation, and enables the selection of ANSYS convergence variables.
- **ANSYS Interface Loads:** This option is only available if an ANSYS Multi-field run is being performed, and enables 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**.

8.1.4.2.4. CFX Plot Line 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. 104)
- [BALANCES](#) (p. 104)
- [FLOW](#) (p. 104)
- [FORCE](#) (p. 104)
- [NEG ACCUMULATION](#) (p. 104)

- [RESIDUAL](#) (p. 104)
- [SOURCE](#) (p. 104)
- [TIMESTEP](#) (p. 104)
- [USER](#) (p. 105)

Further details on the output of the solver are available. For details, see [CFX-Solver Files](#) (p. 33).

8.1.4.2.4.1. ACCUMULATED TIMESTEP

The accumulated timestep values are plotted.

8.1.4.2.4.2. 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.

8.1.4.2.4.3. 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.

8.1.4.2.4.4. FORCE

For details, see [Calculated Wall Forces and Moments](#) (p. 47).

8.1.4.2.4.5. NEG ACCUMULATION

Negative accumulation is the transient term contribution to the balance equation. For details, see [BALANCES](#) (p. 104).

8.1.4.2.4.6. RESIDUAL

For details, see [Residual Plotting](#) (p. 79).

8.1.4.2.4.7. 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 that are defined for the entire domain are automatically computed by the solver, and vary depending on the models that 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.

8.1.4.2.4.8. 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 that 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.

8.1.4.2.4.9. 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 in the CFX-Pre User's Guide](#).

8.1.4.2.4.9.1. 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**.

8.1.4.2.4.10. 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.

8.1.4.2.4.11. ANSYS

The variables that are typically present in 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 Time Step Bisection in the Mechanical APDL Theory Reference .
Line Search Parameter	For details, refer to LNSRCH in the Mechanical APDL Command Reference .
Load Step	In the context of an ANSYS Multi-field run, <code>Load Step</code> is the same as a multi-field timestep or <code>Coupling Step</code> .
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

Variable Name	Description
	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. 81) .
UX, UY, UZ, FX, FY, FZ, TEMP, HFLU	Only in the NLH file. For details, see ANSYS Interface Loads Plots (p. 81) .

8.1.5. 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 causes 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 want to modify (see [Convergence History Plots and User Point 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 want to use as the X-axis variable.

8.1.6. Global Plot Settings Tab

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

For transient runs, this selects or clears plotting of loop data for each inner loop. This is in addition to plotting data for each outer loop/timestep. For details, see [Transient Residual Plotting \(p. 80\)](#).

Note

As turning on the loop 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.

8.2. New Monitor Command

You can use this command to create a new monitor. However, if other properties associated with a workspace need to be defined, use the **Workspace Properties** dialog box instead. Monitors can then be created as required.

- [Workspace Properties Command](#) (p. 99)
- [Monitors Tab](#) (p. 100)

8.3. 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. 135).

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 can.

8.3.1. Stopping 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.

8.3.2. Stopping Runs Using Remeshing

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

8.3.3. Stopping 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 [Mechanical APDL Coupled-Field Analysis Guide in the Mechanical APDL Coupled-Field Analysis Guide](#). In particular, steady state analyses will run until completion, and transient analyses will run until the end of the current time step.

8.3.4. Stopping 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.

8.4. 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 \(p. 21\)](#). To perform the restart, either:

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

8.5. 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. If a solution may be diverging and an intermediate solution must be retained, you should generate backup files.

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.

8.6. Arrange Workspace Command

Arranging the workspace deletes all monitors that are currently showing, regenerates them, and redisplay 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 \(p. 108\)](#).

To arrange the workspace, either:

- Select **Workspace > Arrange**.
- Click *Arrange Workspace* .

8.7. 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* .

8.8. 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 \(p. 109\)](#).

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.

8.9. 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 \(p. 109\)](#).

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 [Defining a Default Plot Monitor \(p. 110\)](#).

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.

8.9.1. Defining a Default Plot Monitor

Rather than creating plot monitors manually, one of the existing ones, `Turbulent.mst` or `workspace-basic.mst`, can be duplicated and modified.

You can then select your modified file from **Workspace > Load Layout** in CFX-Solver Manager.

For example, you could modify a copy of the provided monitor file, `Turbulent.mst`, to include the new variables as follows:

Change these lines in `Turbulent.mst`

```
Variable Rule = CATEGORY = RESIDUAL ; SUBCATEGORY = RMS ; \
EQN_TYPE = list:U-Mom/V-Mom/W-Mom/P-Mass
```

to

```
Variable Rule = CATEGORY = RESIDUAL ; SUBCATEGORY = RMS ; \
EQN_TYPE = list:U-Mom/V-Mom/W-Mom/P-Mass/K-TurbKE/E-Diss.K
```

Note

The monitor file `Turbulent.mst` can be found in the `<CFXROOT>/etc/` directory.

The string for which a search is made can be found from the variable lists in the **Plot Lines** tab of the **Monitors** dialog box. For details, see [Plot Lines Tab \(p. 103\)](#).

8.10. 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 \(p. 79\)](#).

8.11. 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 \(p. 79\)](#).

8.12. 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**.

8.13. Close Workspace 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. 93) or [Monitor Finished Run Command](#) (p. 93), as appropriate.

Chapter 9: CFX-Solver Manager Tools Menu

The **Tools** menu controls layout, plots and text windows that 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:

- 9.1. Edit CFX-Solver File Command
- 9.2. Export Command
- 9.3. Export to ANSYS MultiField Command
- 9.4. Interpolate Command
- 9.5. Edit Run In Progress Command
- 9.6. CCL Propagation in Multi-Configuration Simulations
- 9.7. Edit Current Results File Command
- 9.8. Post-Process Results Command
- 9.9. View Environment Command

9.1. Edit CFX-Solver File Command

The command file section of a CFX-Solver input file can be modified. This enables 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. 83).

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.

9.2. Export Command

If using tools other than CFD-Post for post-processing, data must 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. 149)
- *Running cfx5export from the Command Line* (p. 165)

There must be a results file to reference before exporting.

1. Select **Tools** > **Export Results**.

The **Export** dialog box is displayed.

2. Under **Source** > **Results File**, click *Browse*  and select a results file for export.
3. Under **Source** > **Domain Selection** > **Name**, select the domain to export.

Where multiple domains exist, select either one domain or all domains to export.

4. Set **Timestep**, **Output Only**, **Mesh Options** and **Results Options** as required. For details, see [Generic Export Options](#) (p. 151).
5. Under **Destination** > **Output Format** select CGNS, MSC.Patran, FIELDVIEW, EnSight or Custom User Export.
 - [CGNS](#) (p. 152)
 - [MSC.Patran](#) (p. 155)
 - [FIELDVIEW](#) (p. 158)
 - [EnSight](#) (p. 161)
 - [Custom User Export](#) (p. 163)
6. If required, under **Destination** > **Export File**, click *Browse*  and set the output path and filename.
7. Under **Destination** > `<output_format>` **Options**, configure options as required.

The options are dependent on the **Output Format**. For details, see [Output Format](#) (p. 151).

8. Click **Export**.

Once completed, a message is displayed. Click **OK** to close it.

9.3. 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 Box](#) (p. 149).

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 either one domain or all domains 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 Box](#) (p. 149).

7. Click **Export**.

Once completed a message is displayed. Click **OK** to close it.

9.4. Interpolate Command

ANSYS CFX enables 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 (thereby 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 in the 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 box of the CFX-Solver Manager. For details, see [Run Definition Tab in the CFX-Pre User's Guide](#), [Run Definition Tab in the CFX-Pre User's Guide](#) and [Run Definition Tab \(p. 11\)](#), respectively.

However, if you want 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 box. 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. Optionally, set the precision: under **Executable Settings**, select **Override Default Precision** and choose either **Single** or **Double**.

For details, see [Double-Precision Executables \(p. 141\)](#).

6. If required, under **Interpolator Memory**, adjust the memory configuration. For details, see [Configuring Memory for the CFX-Solver \(p. 17\)](#).
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. 117).

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 that already contains solution data is not recommended. See [Interpolating Onto a Solver Input File with Results Fields in the CFX-Solver Modeling Guide](#) for details.
- Following the steps above, the interpolation is performed in Initial Guess mode. See [Using the CFX-Interpolator in the 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. Optionally, set the precision: under **Executable Settings**, select **Override Default Precision** and choose either **Single** or **Double**.

For details, see [Double-Precision Executables](#) (p. 141).

6. If required, under **Interpolator Memory**, adjust the memory configuration. For details, see [Configuring Memory for the CFX-Solver](#) (p. 17).
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 in the CFD-Post User's Guide](#).

9.4.1. 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 that 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.

9.5. 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 pre-processed for the flow solver. The modified CCL may take several iterations to be updated depending on system load, hardware and problem size.

These 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 \(p. 90\)](#).

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 \(p. 83\)](#).

To access the command file editor, select **Tools > Edit Run In Progress** or click .

Note

The **Edit Run in Progress** command is not selected for simulations involving ANSYS Multi-field couplings or transient blade row modeling. This constraint exists for simulations involving ANSYS Multi-field couplings because of difficulties involved with updating the MFX settings in the ANSYS solver during the run.

9.6. CCL Propagation in Multi-Configuration Simulations

The **Edit Run in Progress** command is selected 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 (such as material properties, expressions, reactions, etc...) are propagated to all subsequent configurations, and
- Changes made to the `FLOW` section of the CCL (such as analysis type, boundary conditions, solver and output controls, etc...) are propagated to all subsequent runs of the current configuration.

9.7. Edit Current Results File Command

Edit the CFX-Solver Results file in the current workspace (if available). This feature is available only when there is a finished CFX-Solver Results file in the current workspace. For details, see [Editing CFX-Solver Input Files](#) (p. 83).

To access the command file editor, select **Tools > Edit Current Results File** or click . The Command File Editor is launched and the current results is opened.

9.8. Post-Process Results Command

Loads the CFD-Post post-processor. For details, see [Overview of CFD-Post in the 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 **Multi-Configuration Load Options** in order to control how the results of a multi-configuration run are loaded, or to load just the last case of such a run.
5. Select or clear **Shut down CFX-Solver Manager**.

If selected, ANSYS CFX-Solver Manager is shut down before CFD-Post is launched.

6. Click **OK**.

9.9. 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: CFX-Solver Manager 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:

11.1. Command-Line Use

11.2. Command-Line Options and Keywords for cfx5solve

11.3. Command-Line Samples

Note

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).

11.1. 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 ANSYS CFX Launcher: **Tools > Command Line**. Alternatively, you can run CFX-Solver Manager from a DOS prompt. For details, see [Command Line in the CFX Reference Guide](#).

The `cfx5solve` command-line options are described in the next section.

11.2. 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
<code>-ansys-arguments <arguments></code>		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. 12) .
<code>-ansys-input <file></code>		For an ANSYS Multi-field run, specifies the Mechanical application input file to use. For details, see MultiField Tab (p. 12) .
<code>-ansys-input-is-complete</code>		For an ANSYS Multi-field run, this turns off the Processing Mechanical input file option. For details, see MultiField Tab (p. 12) .
<code>-ansys-installation <directory></code>		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. 12) .
<code>-ansys-jobname <name></code>		For an ANSYS Multi-field run, sets the jobname for the Mechanical application component of the simulation. The CFX-Solver Manager defaults this to <code>ANSYS</code> . On a restart, the jobname must be the same as for an initial run.

Command-Line Options	Alternative form	Usage
-ansys-license <licensekey>		For an ANSYS Multi-field run, sets the license that the Mechanical application solver should use.
-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. 12).
-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.
-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 that 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.

Command-Line Options	Alternative form	Usage
-check-only		When running in batch mode, this will cause <code>cfx5solve</code> 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 <code>-use-mesh-from-iv</code> option is also specified. Only one <code>-continue-from-file</code> argument can be supplied. See Continuing the History in the 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 (<code>.cfg</code>) file is used unless the <code>-use-mesh-from-iv</code> option is also specified. Only one <code>-continue-from-configuration</code> argument can be supplied. See Continuing the History in the CFX-Solver Modeling Guide for more details.
-cplg-host <port@host>		For an ANSYS Multi-field run, specifies to the CFX-Solver which Port Number and Host Name to use to communicate with the Mechanical application solver. For details, see MultiField Tab (p. 12) .
-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. 93) . Also see the <code>-mdef</code> option.
-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 <code>-single</code> option.
-example <file>	-eg <file>	Starts the CFX-Solver using one of the example CFX-Solver input files provided with the product. The example <code>StaticMixer</code> is currently available.

Command-Line Options	Alternative form	Usage
-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 <code>-interpolate-iv</code> option is also specified. This option has been deprecated and should be replaced by <code>-initial-file</code> or <code>-continue-from-file</code> 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 <code>-interpolate-iv</code> option is also specified. See Continuing the History in the 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 <code>-use-mesh-from-iv</code> option is also specified. See Continuing the History in the 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 enables starting a new run or managing or monitoring an existing run.
-interpolate-iv	-interp-iv	Interpolates the solution from the initial values file, if one is supplied (using the <code>-initial</code> 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 <code>-initial-file</code> or <code>-continue-from-file</code> option, as appropriate.
-interp-double		When running with the solver-based interpolator (<code>-interp-iv</code> option), this option

Command-Line Options	Alternative form	Usage
		will select the double-precision version of the interpolator. It will not override the <code>-interpolator</code> option if both are used.
<code>-interp-single</code>		Uses the single precision ANSYS CFX Interpolator executable.
<code>-interpolator</code> <code><executable></code>		When running with the interpolator (<code>-interp-iv</code> option), this option will start <code><executable></code> instead of the default interpolator.
<code>-job</code>		Keeps the <code>.job</code> file after an ANSYS CFX Solver run. This file contains a brief summary of various solution values, and is most useful for regression purposes.
<code>-job-part</code>	<code>-jobp</code>	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.
<code>-max-elapsed-time</code> <code><elapsed time></code>	<code>-maxet</code> <code><elapsed time></code>	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: <code>-maxet "10 [min]"</code> or <code>-maxet "5 [hr]"</code> .
<code>-mdefinition</code> <code><file></code>	<code>-mdef</code> <code><file></code>	Uses <code><file></code> as the solver input file. This may be a multi-configuration definition file or results file for a restart (that is, <code>.mdef</code> or <code>.mres</code> , 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. 93) .
<code>-mfx-run-mode</code> <code><mode></code>		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. 12) .
<code>-monitor</code> <code><file></code>		When starting ANSYS CFX-Solver Manager, use this option to monitor the run represented by <code><file></code> , which may be a CFX-Solver results file or CFX-Solver output file.

Command-Line Options	Alternative form	Usage
-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.
-norun		Use this option to pre-process the CFX-Solver input file only, without running the CFX-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 pre-processed.
-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>@]<host-name>[:<CFX_ROOT>], if the user name or the ANSYS CFX installation root directory differs from the local host.</user>
-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 Platform 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.

Command-Line Options	Alternative form	Usage
-parallel	-par	Starts the solver in parallel mode. This option can be combined with <code>-partition</code> for a partitioning run. If the <code>-part</code> option is not specified, then the <code>-parfile-read</code> 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. 72) .		
-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 <code>.par</code> .
-parfile-write <parfile>		Specifies the name of a partition file to which to write the information from a partitioning run.
-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. Note that user routines cannot be used in the large memory partitioner.
-part-mode <mode>		Sets the partitioning mode to use when running the partitioner. Valid options are <code>metis-kway</code> (MeTiS k-way), <code>metis-rec</code>

Command-Line Options	Alternative form	Usage
		(MeTiS Recursive Bisection), <code>simple</code> (Simple Assignment), <code>drcb</code> (Directional Recursive Coordinate Bisection), <code>orcb</code> (Optimized Recursive Coordinate Bisection), <code>rcb</code> (Recursive Coordinate Bisection) Finer control over the partitioning method is available through the Command Language.
<code>-part-only</code> <number of partitions>		Starts the solver in partitioning mode only. This is normally equivalent to <code>-part</code> , but may be necessary if partitioning a results file from a previous run.
<code>-part-single</code>		Uses the single precision ANSYS CFX Partitioner. This is the default, but is provided for overriding any information that might be stored in the CFX Command Language dataset in a file from previous runs. Also see the <code>-double</code> option.
<code>-preferred-license</code> <license name>	<code>-P</code> <license name>	Sets the first license used by solver, given the availability of multiple usable licenses.
<code>-priority</code> <level>	<code>-pri</code> <level>	Enables the specification of a job priority to a solver run; the allowed values are <code>Idle</code> (0), <code>Low</code> (1), <code>Standard</code> (2), and <code>High</code> (3). The default value is <code>Standard</code> , which corresponds to a nice increment of 0 on UNIX platforms or a priority level of <code>Normal</code> on Windows platforms. Note that on UNIX platforms, <code>Standard</code> and <code>High</code> job priorities both yield a nice increment of 0.
<code>-save</code>		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.
<code>-serial</code>		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.
<code>-single</code>		Starts the single-precision version of the CFX-Solver, Partitioner, and Interpolator. This is the default, but is provided for overriding any information that might be stored in the CFX Command Language dataset in

Command-Line Options	Alternative form	Usage
		a file from a previous run. Also see the <code>-double</code> option.
<code>-size <factor></code>	<code>-S <factor></code> <code>-s <factor></code>	Changes memory estimates used by the CFX-Solver by a factor of <code><factor></code> . Memory estimates are sometimes inaccurate and this option must be used to increase the memory allocated. For example, using <code>-size 1.05</code> increases memory allocated by 5%. This option provides the same functionality as set in Solver Memory . For details, see <i>Solver Tab</i> (p. 16). Further options for controlling the CFX-Solver memory allocation are available. Execute <code>cfx5solve -help</code> for full details.
<code>-size-cat <size></code>	<code>-scat <size></code>	These flags are for advanced users to change the memory allocation parameters for the solver. Usually, you should use the <code>-size</code> option instead. <code><size></code> is the desired memory allocation in words, and may have K or M appended for <i>kilo</i> or <i>mega</i> . If the suffix is 'x', then the number is treated as a multiplier.
<code>-size-nr <size></code>	<code>-nr <size></code>	
<code>-size-ni <size></code>	<code>-ni <size></code>	
<code>-size-nd <size></code>	<code>-nd <size></code>	
<code>-size-nc <size></code>	<code>-nc <size></code>	
<code>-size-nl <size></code>	<code>-nl <size></code>	
<code>-size-cclsetup <factor></code>	<code>-sizeccl <factor></code>	
<code>-size-cclsetup-cat <size></code>	<code>-scatccl <size></code>	These options are the same as the <code>-size-*</code> options above, but provide sizes needed for the ANSYS CFX CCL Setup executable.
<code>-size-cclsetup-nr <size></code>	<code>-nrccl <size></code>	
<code>-size-cclsetup-ni <size></code>	<code>-niccl <size></code>	
<code>-size-cclsetup-nd <size></code>	<code>-ndccl <size></code>	
<code>-size-cclsetup-nl <size></code>	<code>-ncccl <size></code>	
	<code>-nlccl <size></code>	

Command-Line Options	Alternative form	Usage
-size- cclsetup-nd <size>		
-size- cclsetup-nc <size>		
-size- cclsetup-nl <size>		
-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>	-scatint <size>	These options are the same as the -size-* options above, but provide sizes needed for the ANSYS CFX Interpolator.
-size-interp- nr <size>	-nrnt <size>	
-size-interp- ni <size>	-niint <size>	
-size-interp- nd <size>	-ndint <size>	
-size-interp- nc <size>	-ncint <size>	
-size-interp- nl <size>	-nlint <size>	
-size-mms <factor>	-smms <factor>	
-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>	-sizepart <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 <code>cfx5solve -help</code> for full details.
-size-par <factor>	-sizepar <factor>	

Command-Line Options	Alternative form	Usage
-size-part-cat <size>	-scatpar <size>	These options are the same as the -size-* options, but provide sizes needed for partitioner rather than solver.
-size-part-nr <size>	-nrpar <size>	
-size-part-ni <size>	-nipar <size>	
-size-part-nd <size>	-ndpar <size>	
-size-part-nc <size>	-ncpar <size>	
-size-part-nd <size>	-nlpar <size>	
-size-part-nc <size>	-nlpar <size>	
-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 <code>-solver "linux-amd64/mysolver.exe,linux=linux/mysolver.exe"</code> 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:
		<ul style="list-style-type: none"> • On Unix-like systems, execute <code><CFX-ROOT>/bin/cfx5info -os</code>. • On a Windows system, execute <code><CFX-ROOT>\bin\cfx5info -os</code>.

Command-Line Options	Alternative form	Usage
		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 enables you to use different parallel methods, as listed in the CFX-Solver Manager user interface, instead of the defaults. For parallel runs, you also need to provide the <code>-part</code> or <code>-par-dist</code> 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).

^aLocators for applying physics cannot be modified using the `-cc1` option; they can, however, be changed in CFX-Pre.

11.3. 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

Note

To ensure that the following example works, define the hosts `hosta`, `hostb` and `hostc` in the central `hostinfo.ccl` file. (`cfx5solve` attempts to automatically detect hosts that are not listed in `hostinfo.ccl`, but this is not guaranteed to work.)

To run CFX-Solver in distributed parallel, starting from the CFX-Solver input file `model.def`, and using one partition on `hosta`, two partitions on `hostb`, and four partitions on `hostc`, for a total of seven 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 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-input-is-complete
```

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-slave CFX
```

Pre-process 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 pre-process 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 pre-process 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. Currently the only supported indirect start method is the "Submit to Windows CCS or HPC Queue" method.

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:

- 13.1. Tetrahedral Mesh
- 13.2. Special Partitioner, Solver and Interpolator Executables
- 13.3. Turbulence
- 13.4. Energy Models
- 13.5. CHT Regions
- 13.6. Multicomponent Flows
- 13.7. Multiphase Flows
- 13.8. Additional Variables, Wall Distance Variables, and Boundary Distance Variables
- 13.9. Combustion Modeling
- 13.10. Radiation Modeling
- 13.11. GGI Interfaces
- 13.12. Transient Runs
- 13.13. Mesh Deformation
- 13.14. Bidirectional (Two-Way) Couplings with ANSYS Multi-field

13.1. 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.

13.2. 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* (p. 141)
- *Large Problem Partitioner Executables* (p. 142)

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, and so on.

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; this 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.

13.3. Turbulence

The following topics will be discussed:

- *Zero Equation Model* (p. 142)
- *Two-Equation Models* (p. 142)
- *Reynolds Stress Model* (p. 142)

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 because 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.

13.4. 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.

13.5. 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).

13.6. 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.

13.7. Multiphase Flows

The following topics will be discussed:

- *Homogeneous Model* (p. 143)
- *Inhomogeneous Model* (p. 143)
- *N-Phase Flow* (p. 143)

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.

13.8. 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 in the 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.

13.9. 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 that does not increase linearly with the number of components.

13.10. Radiation Modeling

This adds a single scalar equation. Cost increases are similar to those of the energy equation. For details, see [Energy Models](#) (p. 142).

13.11. GGI Interfaces

An intersection algorithm that 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.

13.12. Transient Runs

Each coefficient loop requires approximately the same CPU time as the equivalent steady-state iteration.

13.13. 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, and so on) 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).

13.14. Bidirectional (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](#), 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×
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 Static-Mixer_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](#).

To post-process 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 that 3rd party software. ANSYS CFX provides predefined translations to a number of different post-processors 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](#).

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 Box \(p. 149\)](#)) or **Tools > Export** (see [Export of Results to Other Formats \(p. 150\)](#)).

This chapter describes:

- 15.1. Export to ANSYS Multi-field Solver Dialog Box
- 15.2. Export of Results to Other Formats
- 15.3. Generic Export Options
- 15.4. Running `cfx5export` from the Command Line
- 15.5. Exporting a Transient Results File
- 15.6. Exporting Particle Tracking Data
- 15.7. Using a Customized Export Program

Alternatively, it is possible to export the results directly from the command line. For details, see [Running `cfx5export` from the Command Line \(p. 165\)](#).

15.1. Export to ANSYS Multi-field Solver Dialog Box

The **Export to ANSYS Multifield Solver** dialog box is used to produce ANSYS `.cdb` files for use in Fluid Structure Interaction cases. To display the dialog box, select **Tools > Export to ANSYS Multifield**.

The sections that follow describe the **Export Options** fields on the dialog box.

15.1.1. 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)
- 2D Stress (element type 154).

Refer to the [Mechanical APDL Theory Reference in the Mechanical APDL Theory Reference](#) for a description of these element types.

15.1.2. 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](#) (p. 150)
- [Offset Values](#) (p. 150)
- [Scale Flow](#) (p. 150)
- [Scale Values](#) (p. 150)

15.1.2.1. Offset Flow

Default Value: 0

Requires a real number corresponding to the offset value applied to solution flows.

15.1.2.2. Offset Values

Default Value: 0

Requires a real number corresponding to the offset value applied to solution values.

15.1.2.3. Scale Flow

Default Value: 1

Requires a real number corresponding to the scaling factor applied to solution flows.

15.1.2.4. Scale Values

Default Value: 1

Requires a real number corresponding to the scaling factor applied to solution values.

15.2. 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 Results** 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.

15.3. Generic Export Options

To configure data for export, the **Export** dialog box must be displayed: select **Tools > Export Results**.

There are numerous export options when writing to the supported 3rd-party formats:

- *Results File* (p. 151)
- *Domain Selection: Name* (p. 151)
- *Timestep Selection: Timestep* (p. 151)
- *Output Format* (p. 151)
- *Export File* (p. 163)
- *Output Only Boundary Geometry and Results* (p. 163)
- *Mesh Options: Use Initial Mesh for Rotating Domains* (p. 164)
- *Results Options: Output Level* (p. 164)
- *Results Options: Include Variables Only Found on Boundaries* (p. 164)
- *Results Options: Use Corrected Boundary Node Data* (p. 164)

15.3.1. Results File

The name of the CFX-Solver results file to be exported.

15.3.2. Domain Selection: Name

The domain or domains in the CFX-Solver results file that are to be exported.

15.3.3. Timestep Selection: Timestep

For transient CFX-Solver results files, the timestep to be exported.

15.3.4. 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.

15.3.4.1. 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.

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.

Note

To configure this option, select **Tools > Export**. The dialog box uses numerous common export options. For details, see [Generic Export Options \(p. 151\)](#).

This section contains:

- [CGNS Options \(p. 152\)](#)
- [Exported Files \(p. 153\)](#)
- [Contents of CGNS Files Written by ANSYS CFX \(p. 153\)](#)
- [Reading Exported Files into a Program Supporting CGNS \(p. 154\)](#)

15.3.4.1.1. CGNS Options

The following options are available when writing a file in CGNS format:

- All options specified in [Generic Export Options \(p. 151\)](#)
- **Geometry Output Only**
 - *Default Value:* Cleared.
 - If selected, only mesh information and boundary condition information is written to the destination file; result variables are not written.
- **Version and File Format**
 - 2.4
The ADF file format will be used in this case.
 - 3.0
You can select either the ADF or HDF5 format.
- 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.
- **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).

- When **Use CGNS Variable Names** is selected ANSYS CFX variable names are mapped to CGNS variable names. For example `Total Pressure` becomes `PressureStagnation`.
- When **Write Full CFX Solver Name as Description** is selected, CFX-Solver will write a description of the full CFX Solver Variable Name. This description is an extra data field that is used by CFD-Post when it exists. This avoids the internal variable names being used when the full CFX Solver name exceeds 32 characters.

Whenever the name of the variable exceeds 32 characters, the internal name for the variable, which is shorter but more cryptic, is written instead. When **Write Full CFX Solver Name as Description** is selected, CGNS files written by CFX-Solver Manager use a new additional data tag that is not subject to the 32-character limit, and that holds the ANSYS CFX Solver Name for each variable. CFD-Post reads the new tag in preference to the old tag, if the new tag exists.

15.3.4.1.2. 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 in ADF format and version 3.0 files in both ADF and HDF5 format. 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.

15.3.4.1.3. 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 user interface. What is seen in the CGNS file reflects what you request. 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. To ensure that the variable names are consistent between CFX and CGNS files, an additional data tag is written for all solution variables, starting in CFX Release 14.0. This tag, named `Ansyz CFX Solver Name`, is exempt from a 32-character limit on variable names, so that the variable names displayed in CFD-Post match those in the results file.

The remainder of this sections describes the data records that are written when creating a CGNS file from ANSYS CFX.

15.3.4.1.3.1. 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.

15.3.4.1.3.2. Zones (`Zone_t`)

A single zone is written under each `Base_t` node for each domain you requested 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.

15.3.4.1.3.3. 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.

15.3.4.1.3.4. 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.

15.3.4.1.3.5. 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.

15.3.4.1.3.6. 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.

15.3.4.1.3.7. ANSYS CFX Connectivity using CGNS for Aerodynamic Noise Analysis

Further information on exporting files is contained in [Aerodynamic Noise Analysis in the CFX-Solver Modeling Guide](#).

15.3.4.1.4. 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.

15.3.4.2. 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. 151).

This section contains:

- [Available Options](#) (p. 155)
- [Exported Files](#) (p. 155)
- [Reading Exported Files into MSC.Patran](#) (p. 156)
- [Exporting Boundary Conditions to MSC.Patran](#) (p. 156)
- [Example Procedure](#) (p. 157)

15.3.4.2.1. Available Options

The following options are available when writing a file in MSC.Patran format:

- All options specified in [Generic Export Options](#) (p. 151)
- **Geometry Output Only (Neutral File)**
 - Default Value: Cleared.

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.

15.3.4.2.2. 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.

15.3.4.2.3. Reading Exported Files into MSC.Patran

You should use the following procedure to import 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>*. You do not need to take any action.

1. Prepare the required PATRAN files.

Run `cfx5export` on the CFX-Solver 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 **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 the 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 message 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. Type 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.

15.3.4.2.4. 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.

15.3.4.2.5. 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 6 \(p. 157\)](#) 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 **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 the 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 message 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/Node1. 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.

15.3.4.3. FIELDVIEW

FIELDVIEW is a stand-alone 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. 151).

This section contains:

- [Available Options](#) (p. 158)
- [Exported Files](#) (p. 159)
- [Reading Exported Files in FIELDVIEW](#) (p. 160)

15.3.4.3.1. 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. 151).
- **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. You should use the most recent version of the file format supported by your FIELDVIEW installation. 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. You should specify the highest version that is supported by the version of FIELDVIEW you have installed.

15.3.4.3.2. 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 \(p. 158\)](#)) 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.

15.3.4.3.3. 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. 160)
- *FIELDVIEW Versions 9 and 10* (p. 160)
- *FIELDVIEW Versions 6, 7, 8* (p. 161)

15.3.4.3.3.1. 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:
 - a. Click **Read Grid or Combined Data**.
 - b. Select the grid file from the file popup and click **OK**.
 - c. Click **Read Results Data**.
 - d. Select the first results file and click **OK**.
 - e. 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:
 - a. Click **Read Grid or Combined Data**.
 - b. Select the combined file from the file popup and click **OK**.

15.3.4.3.3.2. 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:
 - a. Click **Read Grid or Combined Data**.
 - b. Select the grid file from the file popup and click **OK**.
 - c. Click **Read Results Data**.
 - d. Select the first results file and click **OK**.
 - e. 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:
 - a. Click **Read Grid or Combined Data**.
 - b. Select the combined file from the file popup and click **OK**.

15.3.4.3.3.3. 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.

15.3.4.4. 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 \(p. 151\)](#).

This section contains:

- [Available Options \(p. 161\)](#)
- [Export Files \(p. 162\)](#)
- [Reading Exported Files into EnSight \(p. 162\)](#)

15.3.4.4.1. Available Options

The following options are available when writing files for use in EnSight:

- All options specified in [Generic Export Options \(p. 151\)](#)
- **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.

15.3.4.4.2. 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 (and so on)	EnSight Variable Files for scalar variable
<basename>.v01 (and so on)	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.

15.3.4.4.3. Reading Exported Files into EnSight

Files can be read into EnSight 5, 6, 7, and 8 as required.

- [EnSight 8.2 and Later \(p. 162\)](#)
- [EnSight 6, 7, and 8.0 \(p. 163\)](#)
- [EnSight 5 \(p. 163\)](#)

15.3.4.4.3.1. 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.

15.3.4.4.3.2. 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.

15.3.4.4.3.3. 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 **(Set) Geometry**.
2. Click 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.

15.3.4.5. 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. 170). For details on creating a custom export program, see [Creating a Customized Export Program](#).

15.3.5. 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.

15.3.6. 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.

15.3.7. 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 (that is, 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 (that is, 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.

15.3.8. 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](#).

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.

15.3.9. 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 that have values in the interior of the domain(s).

15.3.10. 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 in the CFX Reference Guide](#).

15.4. 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. 171).

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**).

15.4.1. 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 in the 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] [-domain <number>] \
  [-cgns-version <2.4|3.0>] [-HDF5|-ADF] \
  [-geometry] [-help] [-include] [-name <file>] [-summary] \
  [-timestep <number>] [-user <level>] [-norotate] \
  [-boundaries-as-nodes|-boundaries-as-faces] \
```

```
[-one-file-for-transient-cases] [-cgns-names] \
[-no-cfx-variable-description]
[-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>]
```

15.4.2. cfx5export Arguments

A basic description of the cfx5export arguments is given below.

Argument	Usage
Alternative Forms	
-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)
-adf	Write CGNS data as an ADF formatted file.
-ansysfsi	Export data in the ANSYS FSI CDB format.
-ascii	Write the output file in ASCII format, rather than unstructured binary format.
-s	
-binary	Write the output file in unstructured binary format.
-boundary	Boundary data only. Using this argument corresponds to selecting the option Boundary Data Output Only when using cfx5export from CFX-Solver Manager.
-b	
-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).

Argument	Usage
Alternative Forms	
-cgns-version <2.4 3.0>	Write CGNS file as version specified.
-corrected -c	Use corrected boundary node values. This corresponds to enabling Use Corrected Boundary Node Data when using Tools > Export from CFX-Solver Manager.
-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. 151) .
-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.
-eltype	Specifies the ANSYS element type to use for ANSYS FSI CDB file output.
-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 the information in this table.
-hdf5	Write CGNS data as an HDF5 formatted file.
-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.
-no-cfx-variable- description	Don't write a description of the full CFX Solver Variable Name.
-one-file-for-tran- sient-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.

Argument	Usage
Alternative Forms	
-nodes	Use nodes when exporting packet 21 data (groups).
-n	
-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. 164) .
-offset-flow <number>	Offset flows written to ANSYS FSI CDB files by <number>.
-Of <number>	
-offset-val <number>	Offset values written to ANSYS FSI CDB files by <number>.
-Ov <number>	
-patran	Export data for use with MSC/PATRAN.
-regions <region1><,region2>...	Export data for comma-separated list of regions written to ANSYS FSI CDB file.
-REG <version>	Specifies the version used when writing the FIELDVIEW region file.
-scale-flow <number>	Scale flows written to ANSYS FSI CDB files by <number>.
-Sf <number>	
-scale-val <number>	Scale values written to ANSYS FSI CDB files by <number>.
-Sv <number>	
-split-grid-and-results	Use the FIELDVIEW Split Grid and Results file format when writing grid and results files.
-summary	Displays a summary of the domains and timesteps contained in the CFX-Solver Results file.
-f	
-timestep <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.
-time <number>	
-t <number>	
-UNS<version>	Specifies the file format version when writing FIELDVIEW Grid and Results files.
-user <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 <code>cfx5export</code> from CFX-Solver Manager.
-u <level>	
-verbose	Specifying this option may result in additional output being sent to the standard output file (normally the screen).
-v	
<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

Argument	Usage
Alternative Forms	<p><code><file>.v##</code> where ## is the variable number and <code>s</code> indicates a scalar and <code>v</code> a vector unless the <code>-name</code> option is used.</p> <p>Patran geometry will be written to <code><file>.out</code> as a Neutral file, the results template to <code><file>.res_tmpl</code>, and the nodal results to <code><basename>.nod</code> unless the name option is used.</p> <p>Fieldview geometry will be written to <code><file>.fv</code> unless the <code>-name</code> option is used.</p> <p>Note: The files will be created in the same directory as the original files, not necessarily in the current working directory.</p>

15.5. 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. 151)
- [Running cfx5export from the Command Line](#) (p. 165)

15.5.1. 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 1 s, 5 s, and 7 s are exported, they will have the filenames `filename_t1.ext`, `filename_t2.ext` and `filename_t3.ext`.

15.6. 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.

15.7. 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. 170)*
- *Using a Customized Export Program from the Command Line (p. 170)*

15.7.1. 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**.

15.7.2. Using a Customized Export Program from the Command Line

You may run your program directly from the command line started from the ANSYS CFX 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 in the CFX Reference Guide](#).

15.7.2.1. 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`.

15.7.2.1.1. UNIX

To run the program directly on a UNIX system, open a command line started from the ANSYS CFX Launcher (**Tools > Command Line**) and call your program using:

```
./myexport file.res example
```

assuming that the current directory is `/home/smith/export/`.

15.7.2.1.2. Windows

On a Windows machine, run the program by opening a command line started from the ANSYS CFX 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.

15.7.2.2. 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 in the *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 in the *CFX Reference Guide*](#).

Index

A

- Additional Variables
 - CPU and memory requirements of, 144
- ANSYS CFX-Solver Manager
 - customizing, 7
 - starting, 1
- ANSYS GST and NLH variables, 105
- ANSYS multi-field
 - CPU and memory requirements of, 145
- ANSYS Multi-field run, 23
 - restarting, 29
- arrange workspace command, 108

B

- background color
 - setting for Solver monitor, 102
- backup run command, 108
- boundary
 - corrected node values, 167
 - node only data, 167

C

- catalogue size, 17
- CCL file
 - editing, 88
 - structure, 88
- CFD (Computational Fluid Dynamics)
 - job information, 40
- CFX commands
 - cfx5export, 165
 - cfx5interp, 117
- CFX export
 - with transient results, 169
- CFX variables
 - ACCUMULATED TIMESTEP, 104
 - BALANCES, 104
 - FLOW, 104
 - for monitoring a CFX run, 103
 - FORCE, 104
 - NEG ACCUMULATION, 104
 - RESIDUAL, 104
 - SOURCE, 104
 - TIMESTEP, 104
 - USER, 105
- CFX-Solver
 - configuring memory for, 17
 - parallel run, 19
 - pre-processing incomplete configuration files, 137

- run output results, 17
- starting an ANSYS Multi-field Run (FSI), 136
- starting from the command line, 123
- starting in distributed parallel, 136
- starting in local parallel, 136
- starting in parallel, 136
- stopping the CFX-Solver from the command line, 137
- CFX-Solver input file
 - editor - rules, 87
- CFX-Solver input file editor
 - changing appearance of, 87
- CFX-Solver Manager
 - text output window, 5
- CFX-Solver Manager File menu, 93
- CFX-Solver Manager workspace menu commands
 - overview, 99
- CFX-Solver Manager workspace properties command
 - overview, 99
- CFX-Solver Output file
 - absorbed by porous media, 63
 - collected on walls, 63
 - command file, 36
 - completed job information (parallel), 57
 - computed model constants, 45
 - conjugate heat transfer, 58–59
 - continue from last time step (transient only), 63
 - convergence history, 43, 51
 - CPU requirements, 50
 - entered domain, 63
 - exceeded distance limit, 63
 - exceeded integration limit, 64
 - exceeded time limit, 64
 - false transient information, 49
 - fell below minimum diameter, 64
 - final average scales, 50
 - global conservation statistics, 45
 - header, 36
 - host information (parallel), 56
 - initial average scales, 42
 - integrated particle flows, 65
 - integration error, 64
 - job information, 40, 51
 - job information (parallel), 56
 - left domain, 64
 - linear solution, 44
 - maximum residual, 43, 49
 - memory usage, 40
 - memory usage (parallel), 56
 - mesh adaption, 57
 - mesh statistics, 41
 - particle convergence history, 65

- particle fate diagnostics, 63
- particle transport equations, 62
- partitioning information (parallel), 55
- rate, 43
- rigid body, 69
- rms residual, 43
- sliding along walls, 64
- solved equations, 43
- thermal energy (solid), 58–59
- waiting for next time step (transient only), 64
- wall forces and moments, 47
- cfx5cmds command, 88
- cfx5control
 - controls CFX-Solver run, 147
- cfx5export
 - CFX command, 165
- cfx5solve
 - command-line options, 124
 - examples, 135
 - starts CFX-Solver, 123
- cfx5stop
 - stops CFX-Solver, 137
- CHT regions
 - CPU and memory requirements of, 143
- close workspace command, 94
- combustion modeling
 - CPU and memory requirements of, 144
- command file, 36
- command file editor
 - overview, 83
- commands
 - close workspace, 94
- computed model constants, 45
- convergence
 - history, 43, 51
- convergence discontinuities
 - preventing in immersed solids runs, 108
- convergence history plots
 - for CFX-Solver Manager runs, 2
 - printing, 3
- corrected boundary node values, 167
- CPU requirements, 50

D

- define run command, 93
- Define Run dialog box, 10
- distributed parallel
 - setup, 20
- double buffering, 97
- double-precision executables
 - CPU and memory requirements of, 141

E

- energy models
 - CPU and memory requirements of, 142
- EnSight
 - options, 161
 - post-processing, 149
 - using exported files, 162
- equation residual, 79
- export
 - boundary conditions to MSC/Patran, 156
 - cfx5mondata, 4
 - executable, 170
 - plot data, 4
 - transient results file, 169

F

- false transient information, 49
- fieldview
 - post-processing, 149
- file
 - format, 169
- file export utility, 149
- Fluid Structure Interaction cases
 - exporting data for, 149

G

- GGI interfaces
 - CPU and memory requirements of, 144

H

- homogeneous model
 - CPU and memory requirements of, 143

I

- immersed solids runs
 - preventing convergence discontinuities, 108
- importing results
 - to Fieldview, 160
- information
 - false transient, 49
- inhomogeneous model
 - CPU and memory requirements of, 143
- interpolating results, 115
- Interpolator tab, 16

L

- large problem partitioner executables
 - CPU and memory requirements of, 142
- load layout command, 109
- local parallel
 - setup, 20

M

- Memory Management System (MMS)
 - catalogue size, 17
- mesh
 - statistics, 41
- mesh deformation
 - CPU and memory requirements of, 144
- mesh interpolation, 115
 - using command line, 117
- monitor finished run command, 93
- monitor points
 - plotting, 105
- monitor run in progress command, 93
- monitors
 - deleting, 101
 - general settings, 101
 - modifying, 101
 - plot lines, 103
 - range settings, 102
 - setting background color, 102
- mouse mapping, 97
- MSC/Patran
 - options, 155
 - post-processing, 149
- MST file
 - defines a plot monitor, 110
- multi-configuration Run History page, 6
- multi-field run, 23
 - restarting, 29
- multicomponent flows
 - CPU and memory requirements of, 143
- MultiField tab, 12
- multiphase flows
 - CPU and memory requirements of, 143

N

- N-phase flow
 - CPU and memory requirements of, 143
- new monitor command, 107

O

- options
 - ANSYS CFX-Solver Manager, 95
 - common, 96

P

- parallel run
 - configuring, 19
 - of CFX-Solver, 19
- parallel run job information, 57
- particle transport

- particle fates, 63
- partition file
 - producing, 135
- Partitioner tab, 13
- partitioning
 - coupled, 14
 - independent, 13
 - information, 55
 - viewing, 73
- plot monitor
 - creating a new, 110
 - shows expression values vs timestep, 100
- post-processing
 - EnSight, 149
 - fieldview, 149
 - MSC/Patran, 149
- printing
 - solver convergence history, 3
- priority
 - specifying a renice value, 147

R

- radiation modeling
 - CPU and memory requirements of, 144
- renice
 - adjusting the priority of a CFX-Solver run, 147
- reset to default workspace command, 110
- reset-priority
 - adjusting the priority of a CFX-Solver run, 147
- residual monitor
 - shows contents of text files updated during the run, 100
 - shows residual values vs timestep, 100
- residual plot
 - obtaining for old runs, 80
- residual plotting
 - overview, 79
 - transient, 80
- restart current run command, 108
- restarting a run, 21
- results
 - interpolating, 115
- results file, 69
- results files
 - backing up, 108
 - editing, 83
- Reynolds Stress model
 - CPU and memory requirements of, 142
- rigid body
 - output file, 69
- rules file, 89
- Run Definition tab, 11

runs
 monitoring, 107
 preventing discontinuities, 108
 restarting current, 108
 stopping current, 107

S

save layout command, 109
Save Settings, 9
solver run
 Interpolator tab, 16
 MultiField tab, 12
 overview, 9
 Partitioner tab, 13
 restarting, 21
 Run Definition tab, 11
 Solver tab, 16
 using mesh adaption, 22
Solver tab, 16
stop current run command, 107
surface charge, 67

T

tetrahedral mesh
 CPU and memory requirements of, 141
text output window
 in CFX-Solver Manager, 5
toggle layout type command, 108
transient information
 false, 49
transient runs
 CPU and memory requirements of, 144
transient simulations
 exporting data from, 169
two-equation models
 CPU and memory requirements of, 142

U

user
 export, 163

V

variables
 plotting by a specific, 106
variables file, 89
view MAX residuals command, 110
view RMS residuals command, 110

W

Workspace selector
 for CFX-Solver Manager runs, 2

Z

zero-equation model
 CPU and memory requirements of, 142
zombie processes
 terminating, 56