Managing Your CFD run

Introduction

This chapter describes the CFX-5 Solver Manager, which is used to control the CFX-5 Solver as it solves your CFD problem. It provides an interface which allows you to set some attributes for your CFD calculation and control the CFX-5 Solver interactively as the solution develops. The CFX-5 Solver may also be run in “batch mode” (with no graphical user interface), which is particularly useful for large calculations, and this is described at the end of this chapter.

The CFX-5 Solver is the module of CFX-5 which actually calculates the solution to your CFD problem. The CFX-5 Solver Manager provides a graphical user interface to the CFX-5 Solver in order to give you information about the emerging solution, and to provide you with an easy way to control the CFX-5 Solver (which can also be run separately - see **Starting the CFX-5 Solver from the Command Line (p. 68)**).

To start the CFX-5 Solver Manager, see **Starting the CFX-5 Solver Manager (p. 4)**.

Overview of the CFX-5 Solver Manager documentation

The documentation which is available on the CFX-5 Solver Manager can be divided into two categories. Firstly, there is the documentation which is specific to CFX-5 Solver Manager and can be found in this chapter:

- **Starting the CFX-5 Solver Manager p. 4**
- **The CFX-5 Solver Manager Main Menu Bar p. 11**
- **How to use the CFX-5 Solver Manager p. 48**
- **Residual plotting p. 65**
- **Starting the CFX-5 Solver from the Command Line (p. 68)**.

Other relevant documentation can be found in:

- **CFX-5 Solver Files (p. 101)**
- **The File Export Utility (p. 139)**.
- **CFX-5 Solver (p. i)**.

There is also documentation which is more general, but includes details relevant to the CFX-5 Solver Manager. This includes:

- **Introductory Tutorials (Ch. in CFX-5 Tutorials)** and **Advanced Tutorials (Ch. in CFX-5 Tutorials)**.
- **Frequently Asked Questions** (in **Introduction to CFX-5**).
- **Glossary** (in **Introduction to CFX-5**).

Installation and machine-specific details can be found in:

- **Installing CFX-5**.

This chapter describes the layout and functions of the Solver Manager in the first sections, then gives more specific details for use of the Solver Manager towards the end of the chapter.

Starting the CFX-5 Solver Manager

The CFX-5 Solver Manager can be started from within CFX-Build, or directly from the command line.

Starting the CFX-5 Solver Manager from within CFX-Build

To start the CFX-5 Solver Manager from CFX-Build, you should select **Start Solver Manager** when writing the model Definition File using the **Definition File** form in CFX-Build. The CFX-5 Solver Manager is passed the Definition File name automatically. If this Definition File does not need an Initial Values File, then this is sufficient to define a Run and you can start the CFX-5 Solver immediately by pressing the **Start Run** button on the CFX-5 Solver Manager. However, if your Definition File needs the specification of an Initial Values File, you will need to tell the CFX-5 Solver Manager which file these values can be found in by using the **File>Define** menu option. See **How to use the CFX-5 Solver Manager (p. 48)** for more details.

You may also choose to start the CFX-5 Solver Manager from the **Tools** menu in CFX-Build. In this case, no Definition File information is passed to the CFX-5 Solver Manager, and you will have to define a run using the **Define Run** form before being able to run the CFX-5 Solver.

Starting the CFX-5 Solver Manager from the command line

You may start the CFX-5 Solver Manager by typing

```
cfx5solve -interactive
```

followed by the Return key, into a UNIX terminal window or a Windows command window which is set up correctly to find the CFX-5 commands. In this case, no Definition File information is passed to the CFX-5 Solver Manager, and you will have to define a run using the **Define Run** form before being able to run the CFX-5 Solver. More details about additional command line options can be found in **Starting the CFX-5 Solver from the Command Line (p. 68)**.

Starting the CFX-5 Solver Manager from the CFX-5 Launcher

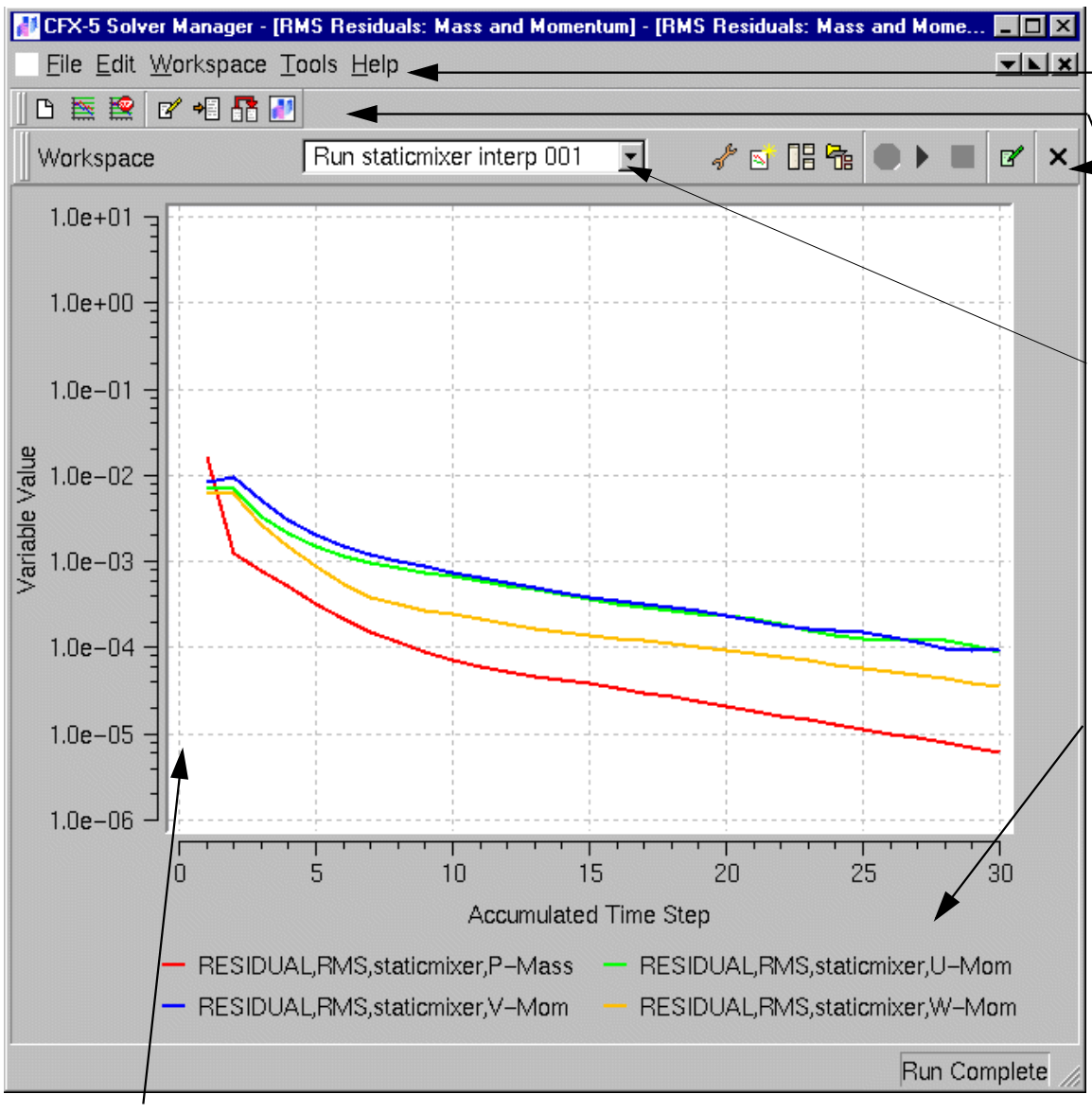
To run the CFX-5 Solver Manager from the CFX-5 Launcher, set the working directory correctly and then click on the **CFX-5 Solver Manager** button - see **CFX-5 Launcher (p. 255 in Installation & Introduction to CFX-5)** for more details. In this case, no Definition File information is passed to the CFX-5 Solver Manager, and you will have to define a run using the **Define** form before being able to run the CFX-5 Solver.

Solver Manager Objects

The Graphics Window.

The Graphics Area displays the residuals for the CFD calculation at each timestep. The variables plotted in the window can be selected from the Workspace menu (please see **Workspace (p. 26)**) Any number of graphics windows can also be created.

You can save or print an image of the graphics window (**Printing and Saving Images (p. 20)**), or export data from the plot monitor to view in another application (**Exporting Plot Data (p. 22)**).



Main Menu Bar

Toolbars

The workspace selector shows the name of the current run in the display and allows you to change to show other runs.

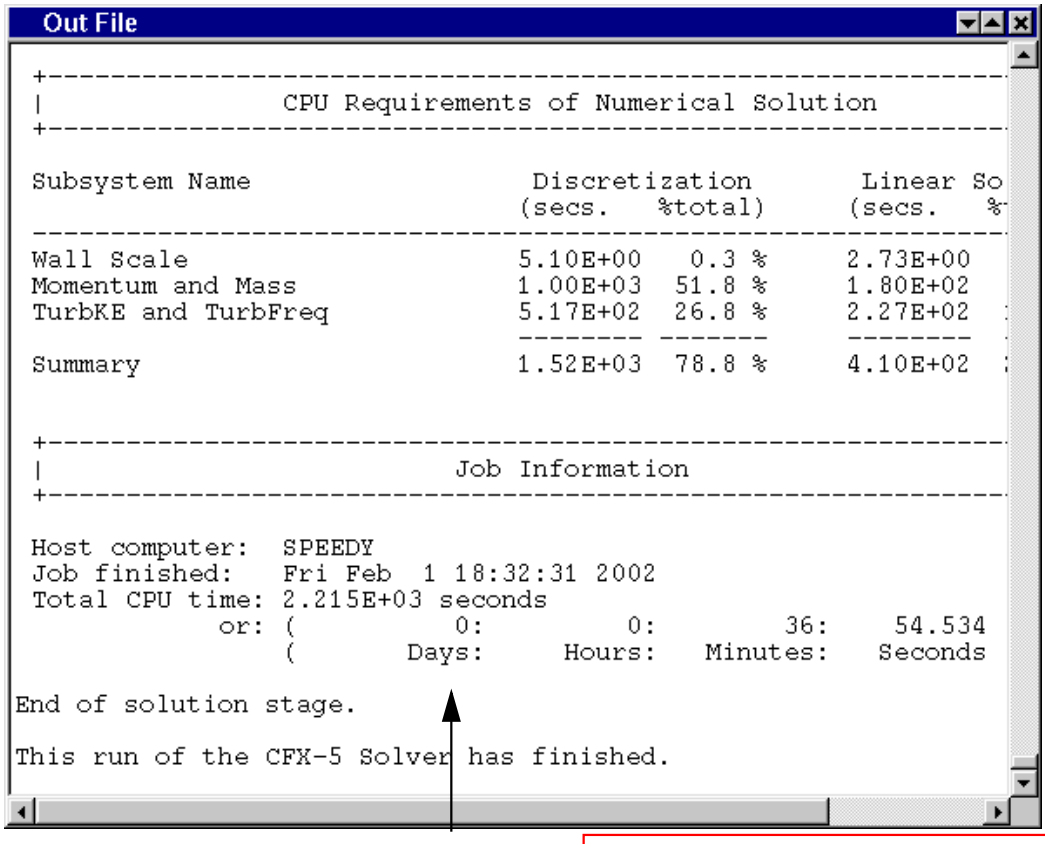
The legend shows the variable associated with each coloured line visible in the graphics window

The main area of the graphics window shows the value of each plotted variable at each timestep. The legends on each scale give an indication of the progress of convergence. Each and every variable pertinent to the solution can be made visible or invisible. Please see **Workspace (p. 26 in CFX-5 Solver and Solver Manager)** for more details.

More Help:

- **The Graphics Window. (p. 5 in CFX-5 Solver and Solver Manager)**
- **Customising the Solver Manager (p. 9 in CFX-5 Solver and Solver Manager)**
- **How to use the CFX-5 Solver Manager (p. 48 in CFX-5 Solver and Solver Manager)**

The Text Window



The text window shows information about the simulation and how the solution is proceeding. This information is also written the the OUT file in the Working Folder. See **The CFX-5 Output File (p. 106 in CFX-5 Solver and Solver Manager)** for detail on the contents of the OUT file.

More Help:

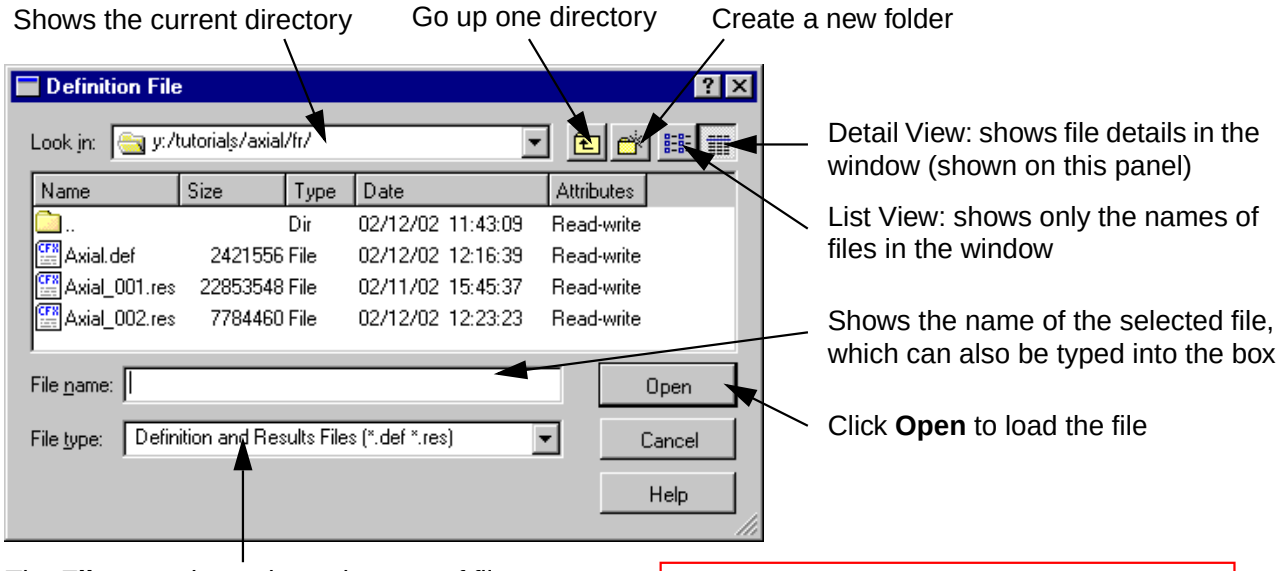
- **The Text Window (p. 7 in CFX-5 Solver and Solver Manager)**
- **Status of a Run (p. 19 in CFX-5 Solver and Solver Manager)**
- **Residual plotting (p. 65 in CFX-5 Solver and Solver Manager)**

The Text Output window informs you of the progress of your solution. This text is also written to the CFX-5 Output File, which is described in **The CFX-5 Output File (p. 106)**. If you wish to save the current state of the output window (even when a run is in progress), right-click in the window and select **Save As**. Enter the name of the text file and click **Save** to save the file. You can also search for any words within the text field by right-clicking in the window and selecting **Find**.

Common Panels and features

Loading files

Most panels have been individually screen captured in this section of the documentation. There are a number of situations requiring loading of files where slightly different panels appear. To avoid unnecessary repetition of the same instructions, documentation for these panels will appear once here.



The **File type** depends on the type of file you are loading. For example, when loading a results file the file type is set to **`.res`** by default. This filters the files and shows only **`.res`** files in the window. **All Files** can also be selected to show every file in the named directory.

☞ **More Help:**
- **Loading files (p. 8 in CFX-5 Solver and Solver Manager)**
- **Common Panels and features (p. 8 in CFX-5 Solver and Solver Manager)**
- **The CFX-5 Solver Manager Main Menu Bar (p. 11 in CFX-5 Solver and Solver Manager)**

Cancel, Restore (or Reset) and Defaults buttons

These buttons, which are found on several of the forms in the CFX-5 Solver Manager, perform the same functions as in CFX-Post, and are explained here.

Cancel Button

The **Cancel** button will quit an operation that you are carrying out. Clicking **Cancel** when loading a file will close the panel and return you to the previous panel or window.

Restore

Clicking **Restore** will return the settings to those stored in the database (i.e to those committed last time **Apply** or **OK** was pressed).

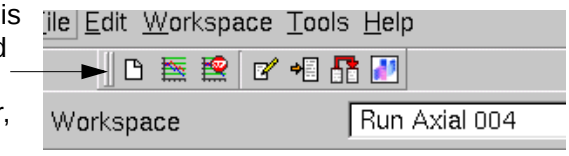
Defaults

Clicking on **Defaults** will return you to the default settings of the CFX-5 Solver Manager.

Customising the Solver Manager

As well as having full control over the size and position of all windows, (which can be moved and sized in the same way as standard windows) you can also move the toolbars to convenient positions by dragging and dropping them.

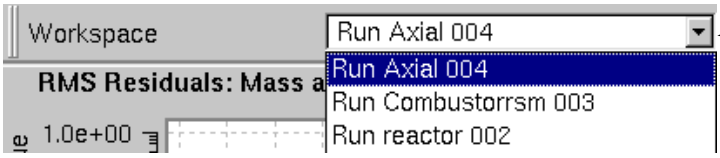
Moving, minimising and maximising toolbars is very simple. To move a toolbar, click and hold on the left edge while dragging to a new position. To minimise or maximise the toolbar, click once on the left edge.



Dragging a toolbar to the edge of the screen can allow you to change its orientation. As an example, the toolbar above can easily be moved to a vertical position on the left of the screen by dragging the toolbar over to the left edge of the screen.

The Workspace menu allows you to create and modify text and graphics windows. For example, you may wish to study the convergence of a variable not currently visible in the graphics window. You can toggle the visibility of all variables on and off, allowing for maximum flexibility.

When studying more than one run, switching between them can be done extremely quickly and easily using the workspace selector.



As well as displaying the current run, the workspace selector allows you to change it. Click on the arrow to select a different run.

A number of templates are also available for the Solver Manager which allow you to load a predefined configuration of graphics and text windows. For example, if you are carrying out a run in which takes into account turbulence and heat transfer, you may simply select the **Turbulent Energy** layout, (which shows three graphics windows of momentum, turbulence and energy along with a text window) without needing to create the windows manually. You may also create your own layouts and load them when required. Please see **Creating a Custom User Layout (p. 36)**. For full details on how to use the Workspace menu features, please see **Workspace (p. 26)**

The CFX-5 Solver Manager Main Menu Bar

The main menus can be found along the top menu bar of the Solver Manager. Detailed descriptions of functions associated with each menu are described in this section.

File

The **File** menu has the following options.

A summary of what each option will do is given below. This is followed by a detailed description of each one.

Define Run... The **Define Run** option is used for setting up a Run of the CFX-5 Solver. It appears automatically when the CFX-5 Solver Manager is launched with the name of a Definition File passed as an argument. This panel is also used to specify an Initial Values File, set Parallel and Partitioning parameters and set some Solver options for the run. See below for more details on **Define Run (p. 11)**.

Monitor Run in Progress This is used when the Solver Manager has been closed and you wish to re-open the display of a run currently underway in the solver..

Monitor Finished Run Used when you wish to view the .out file of a finished run. The way in which the run progressed is plotted in the graphics area and all of the values for each timestep are reproduced in the text area.

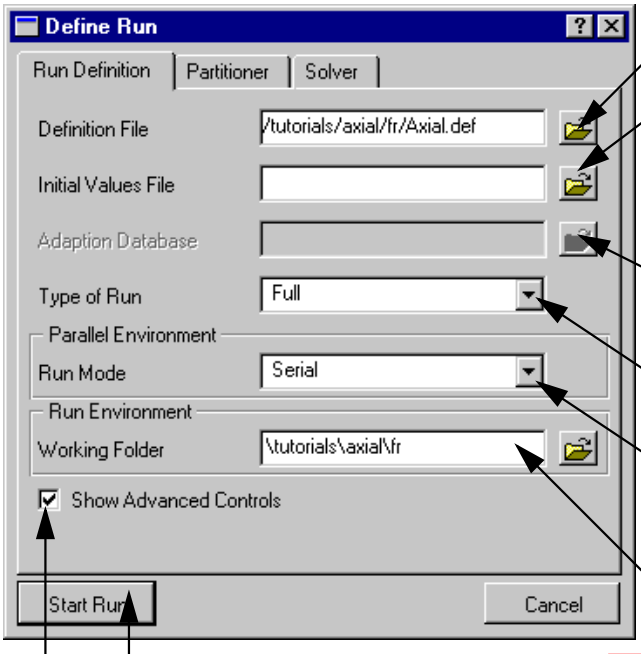
Quit Exits the CFX-5 Solver Manager. See **Quit (p. 20)**.

Define Run

Running the CFX-5 Solver involves starting the CFX-5 Solver, passing it the geometry, models, boundary conditions and start-up information that it needs, and letting it attempt to calculate a solution to your CFD problem. The **Define Run** form is where you specify the information

which is passed to the CFX-5 Solver. Some Runs of the CFX-5 Solver only require you to tell it the name of the CFX-5 Definition File; others will also require the name of an Initial Values File (see **When do you need to specify an Initial Values file? (p. 48)**).

When opened, the **Define Run** form looks like the form shown below (extra options will be present if your run is a parallel run). Extra tab menus become available when Show Advanced Controls is switched on. Each of these tab menus is explained below.



Browse to your definition file by clicking here.

If you require an Initial Values file, browse to it by clicking here. See **When do you need to specify an Initial Values file? (p. 48 in CFX-5 Solver and Solver Manager)** for more information.

If you are using a database for mesh adaption, click here to browse to the location.

Choose the type of run. Options are **Full** and **Partitioner Only** (for parallel runs)

Select whether your run is **Serial**, **Local Parallel** (your own machine only) or **Distributed Parallel**.

The Working Folder is the directory to which all ouput files from the Solver will be written.

After setting the solver parameters, click here to begin the solution process

This toggle makes the other tab menus, **Partitioner** and **Solver** visible.

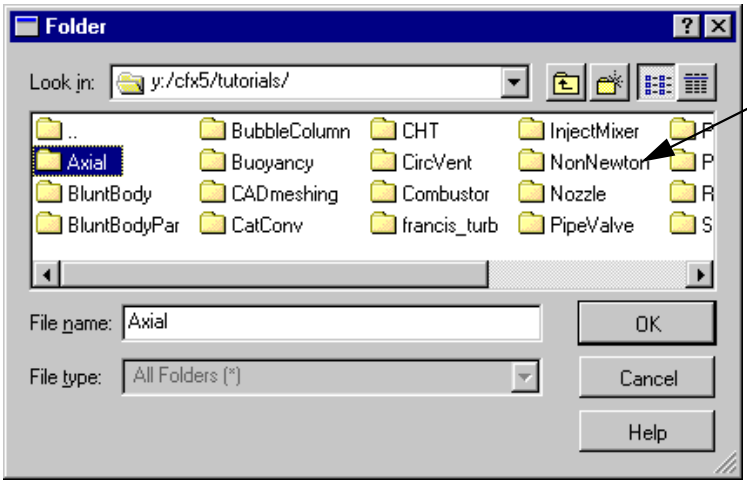
More Help:

- **Define Run (p. 11 in CFX-5 Solver and Solver Manager)**
- **Using the CFX-5 Solver Manager for a Serial Run (p. 49 in CFX-5 Solver and Solver Manager)**
- **Setting Up and Running a Parallel Run (p. 52 in CFX-5 Solver and Solver Manager)**
- **Partitioning the model only (p. 54 in CFX-5 Solver and Solver Manager)**
- **File (p. 11 in CFX-5 Solver and Solver Manager)**
- **How to use the CFX-5 Solver Manager (p. 48 in CFX-5 Solver and Solver Manager)**

Run Definition

The information required to complete this form varies depending on the application.

1. Click on the icon to specify the Definition File.
2. If your Definition File requires you to specify an Initial Values File, then you should specify this in the same way as you specify a Definition File, in the **Initial Values File** box.
3. If you are running in parallel and have already partitioned your run, you may also select the partition file here. by clicking on the icon next to **Initial Partition File**. (this will be called `<filename>.par` if your Definition File was called `<filename>.def`).
4. If you are using grid adaption, identify the CFX-5 database file under **Adaption Database**.
5. The Type of Run box specifies whether a complete solver run will be carried out (**Full**), or whether a partition will be carried out without running the solver (**Partitioner Only**).
6. To run the solver in Parallel, select an option from the **Run Mode** box. Please see **Setting Up and Running a Parallel Run (p. 52)** for more details.
7. The **Working Folder** is used to specify the current working directory. The files that are produced during a solver run will be written to the directory specified in this box. The directory that appears by default is the directory from which the Solver Manager was launched.



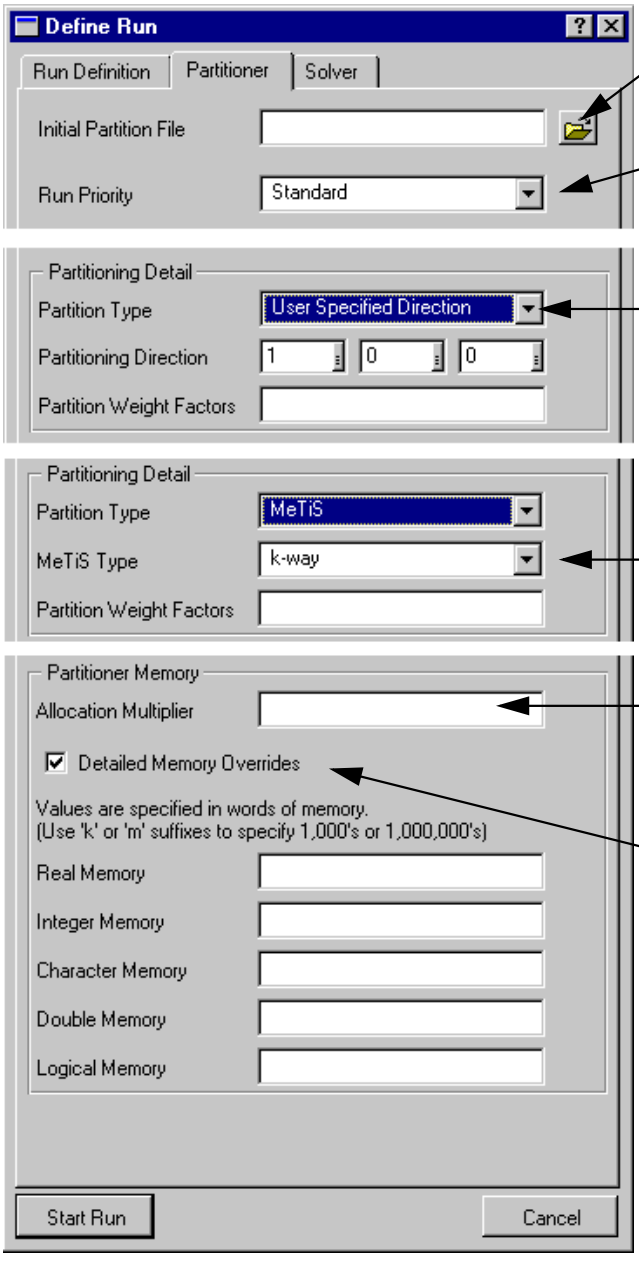
Browse to the directory in which you wish to write all files created by the solver and then click **OK**. For more details on the panel in general, please see **Loading files (p. 8 in CFX-5 Solver and Solver Manager)**

More Help:

- **Loading files (p. 8 in CFX-5 Solver and Solver Manager)**
- **Run Definition (p. 12 in CFX-5 Solver and Solver Manager)**
- **File (p. 11 in CFX-5 Solver and Solver Manager)**

Partitioner

Only reference this section if you are carrying out partitioning of the mesh. The form below is a composite image including more than one option for **Partition Type**.



If you have already partitioned your mesh, you can load the partition file (with the extension `.par`) by clicking here.

Set a run priority for partitioning. Options available are **Low**, **Standard** and **High**..

Choose a partition type from **MeTis**, **Recursive Co-ordinate Bisection** and **User Specified Direction**.
For User Specified Direction a **Partitioning Direction** must be set and option **Partition Weight Factors** can be set.
For Recursive Co-ordinate Bisection no further options need to be specified.
For a MeTiS partition, the **MeTis Type** can be either **k-way** or **recursive bisection**. Optional **Partition Weight Factors** can be set.
See **Partitioning (p. 505 in CFX-5 Solver and Solver Manager)** for details.

If you encounter problems with insufficient memory, an **Allocation Multiplier** can be set to increase memory allocation (e.g. a value of 1.1 increases memory allocation by 10%).

Detailed memory settings can be altered here. Please see **Partitioner (p. 14 in CFX-5 Solver and Solver Manager)** for more details.

More Help:

- **Partitioner (p. 14 in CFX-5 Solver and Solver Manager)**
- **Partition Weight Factors (p. 15 in CFX-5 Solver and Solver Manager)**
- **Define Run (p. 11 in CFX-5 Solver and Solver Manager)**
- **How to use the CFX-5 Solver Manager (p. 48 in CFX-5 Solver and Solver Manager)**

1. If you are using a mesh that has already been partitioned, you can select the file in the **Initial Partition File** box. You can also view the partitions prior to running the CFX-5 Solver, see **The CFX-5 Partition File (p. 126)**.

2. The **Run Priority** can be set to **Low**, **Standard** or **High** for the partition process.

3. Next set the **Partition Type**. If you are new to running in parallel, we recommended you use **MeTiS** as the **Partition Type**, the default. See **Multilevel Graph Partitioning Software - MeTiS (p. 506 in CFX-5 Solver and Solver Manager)** for more information. If you select **User Specified Direction** (which allows you to specify in which direction the partition takes place) then you will be asked to input a vector to specify the direction of the partition (see **User Defined Direction (p. 507 in CFX-5 Solver and Solver Manager)** for more details). You may also choose Recursive Coordinate Bisection, see **Recursive Coordinate Bisection (p. 507 in CFX-5 Solver and Solver Manager)** for more details.

Partition Weight Factors

The Partition Weight Factors widget takes a list of comma-separated numbers, ie.:

`2, 2, 1.5, 1` (for 4 partitions)

If used, there should be an equal number of entries in this list as there are partitions. For a distributed run like the following:

Host	# of Partitions
foo	1
bar	2
baz	1

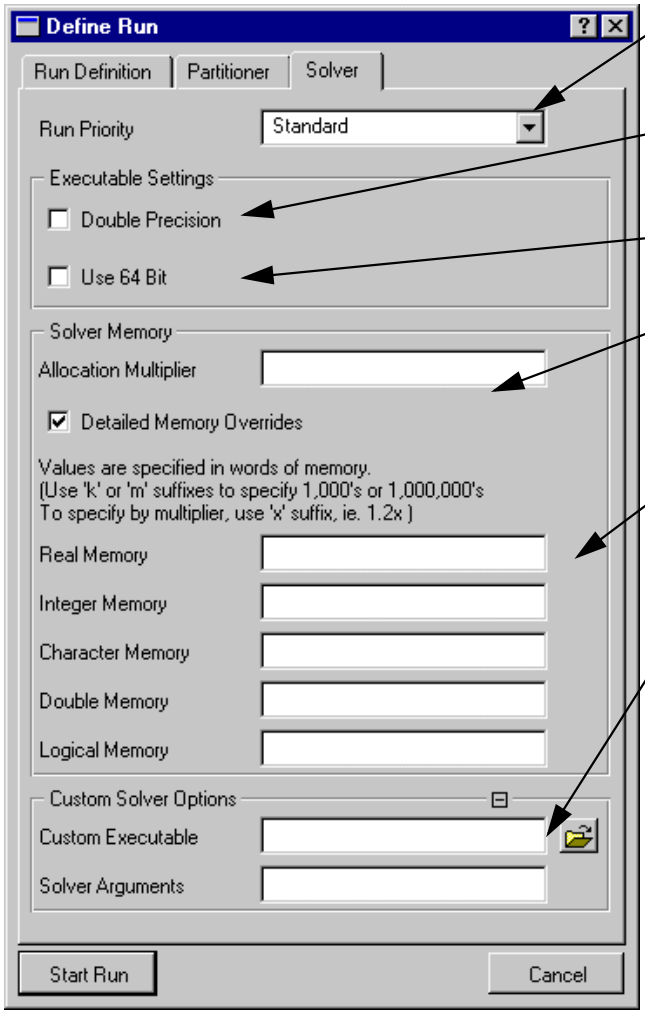
the above weighting list affects the partitions in order (ie. weight for foo = 2, weight for bar's partitions are 2 and 1.5, and weight for baz is 1.)

4. Depending on the partioning method you have chosen, the form will change to set further options.

5. If you wish to configure the memory settings, you may enter an **Allocation Factor** to increase or decrease the amount of Partitioner Memory. You should only need to use it if the CFX-5 Solver fails with an “Insufficient memory” message. In most cases, you should then only need to increase the memory by a few per cent i.e. specifying a Allocation Factor of 1.05. You are advised not to set this factor to less than 1, since it is unlikely that the CFX-5 Solver would then have enough memory to complete the partition.

6. If you require further memory control, for example if you need a lot of one particular type of memory, you can select the **Detailed Settings** toggle. This allows you to explicitly specify the amount of Real, Integer, Double, Character and Logical memory to use. You should type a number into the data entry box corresponding to the number of words of memory, then select either **M K** or **X** for mega-words, kilo-words or a straight memory multiplier respectively (i.e. **2 X** doubles the current memory allocation, and **15 M** would be 15 million words of memory. When setting a Detailed Specification, if a value is not specified for a particular type of memory, the value calculated by the Solver will be used. If a value is entered, it will override the automatic estimate made by the Solver.

Solver



The **Run Priority** can be set to **Low**, **Standard** or **High**.

Click to toggle **Double Precision** (64 bits is used for floating point numbers).

Check to enable the 64 bit solver executable. To use this feature, there must be a 64 bit solver executable available for your platform.

If you encounter problems with insufficient memory, an **Allocation Multiplier** can be set to increase memory allocation (e.g. a value of 1.1 increases memory allocation by 10%).

Help on **Detailed Settings** for memory can be found in **Solver (p. 17 in CFX-5 Solver and Solver Manager)**

If you are using a custom solver executable, you can select its location. Any command line arguments that must be supplied to the program can be entered in the Solver Arguments box.

More Help:

- **Solver (p. 17 in CFX-5 Solver and Solver Manager)**
- **Define Run (p. 11 in CFX-5 Solver and Solver Manager)**
- **How to use the CFX-5 Solver Manager (p. 48 in CFX-5 Solver and Solver Manager)**

1. In the Run Priority box, choose a value from **Low**, **Standard** or **High**. You may also specify the priority with a command line argument. This command can be found in the **command line table** on page **71**.

2. To activate **Double Precision**, check the box. Double Precision is available on Windows, Compaq, IRIX, and Linux platforms. For more information please see **Double Precision (p. 61)**. To view the platform support for double precision, please refer to **Supported Platforms (p. 29 in Installing & Introduction to CFX-5)**.

3. To enable the 64 bit executable, check the 64 bit toggle. A 64 bit solver executable must exist for your platform in order to use this feature, and you must be running on a 64 bit machine. For more details, please see **64-bit (p. 60)**. To view the platform support for 64-bit, please refer to **Supported Platforms (p. 29 in Installing & Introduction to CFX-5)**

4. If you wish to configure the memory settings, you may enter an **Allocation Factor** to increase or decrease the amount of Solver Memory. You should only need to use it if the CFX-5 Solver fails with an “Insufficient memory” message. In most cases, you should then only need to increase the memory by a few per cent i.e. specifying a Allocation Factor of 1.05. You are advised not to set this factor to less than 1, since it is unlikely that the CFX-5 Solver would then have enough memory to complete the calculation.

5. If you require further memory control, for example if you need a lot of one particular type of memory, you can select the **Detailed Settings** toggle. This allows you to explicitly specify the amount of Real, Integer, Double, Character and Logical memory to use. You should type a number into the data entry box corresponding to the number of words of memory, then select either **M**, **K** or **X** for mega-words, kilo-words or straight multiplier respectively. So **15 M** would be 15 million words of memory and **`1.2 X`** would increase the memory by 20%. When setting a Detailed Specification, if a value is not specified for a particular type of memory, the value calculated by the Solver will be used. If a value is entered, it will override the automatic estimate made by the Solver.

The New Run will be given a name which consists of the Definition File name (with the `.def` removed) followed by a three-digit number. This number is the lowest number available that does not result in the name duplicating that of another Run. Normally it will be `001` for the first Run using that Definition File. For instance, if your Definition File is called `buoyant.def`, the Run name will normally be `buoyant_001`, if it is the first Run you have defined using this Definition File. When you first create a new Run, its name is immediately entered under **Current Run** at the top of the form, and its status is **New**.

After closing the **Define Run** form, you can only control the run which is indicated in the Workspace box. If, at any time, you want to start a new run, you can always come back to the **Define Run** form, and change the **Run Definition**. You can make any other started runs current by selecting File>Monitor Current Run and choosing the .out file for that run. You can then easily switch between runs by selecting which run you would like to see from the **Workspace** box.

You should note that if you select an **Input File** which is not in the current directory, the CFX-5 Solver will write the Results File and Output File to the directory which was current when you started the run, and not to the same directory as the **Initial Values File**. The current directory is that directory which is specified under **Working Folder**, and it is that directory which the CFX-5 Solver works in, and puts the generated files in.

You may not interact with other menus in the CFX-5 Solver Manager while the **Define Run** form is up.

See **How to use the CFX-5 Solver Manager (p. 48)** for a step-by-step description on using the CFX-5 Solver Manager.

Status of a Run

While the CFX-5 Solver is actually performing calculations upon on Run, it creates a temporary directory called `<name>.dir`, where <name> is the Run name.

Monitor Run in Progress

To examine a run that is currently being solved, select **Monitor Run in Progress** from the File Menu or click on the  icon. You can browse to the temporary directory < >.dir and the data up to the current timestep will be loaded.

Monitor Finished run

To examine a run that has finished, select **Monitor Finished Run** from the File Menu or click on the  icon. You can then browse to the .res file and the data will be loaded. You may also view error (.err) and out (.out) files to investigate why a run failed.

Quit

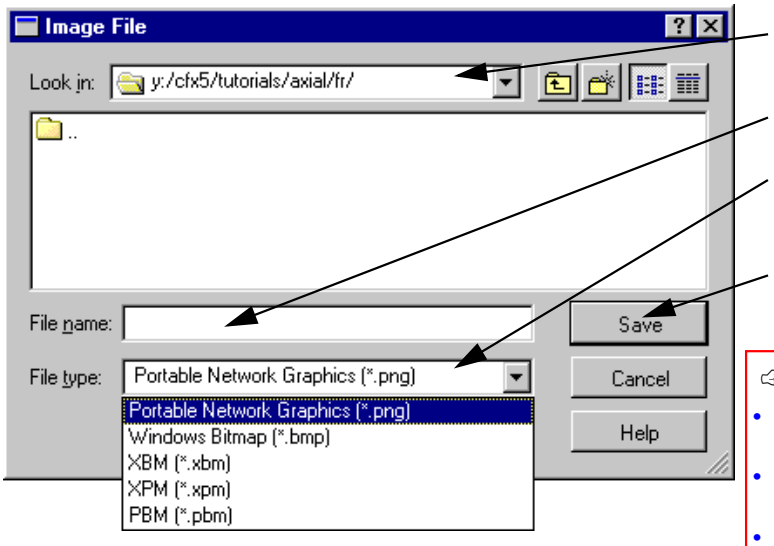
If you select **File>Quit**, the Solver Manager will close. There will be a chance to confirm that you really do want to close it. However, closing the CFX-5 Solver Manager will not stop the CFX-5 Solver jobs which are currently running, and you can reopen the CFX-5 Solver Manager and take control of these jobs again simply by opening the CFX-5 Solver Manager and selecting Monitor Run in Progress (see above).

Printing and Saving Images

Saving an Image

To save an image of the graphics window, simply right-click on the window and select **Save as Image**. The following form appears.

.



Browse to the directory in which the image will be saved.

Enter a name for the image.

The File Type: box lists the available formats for saving the image.

Click Save to write the image file to disk.

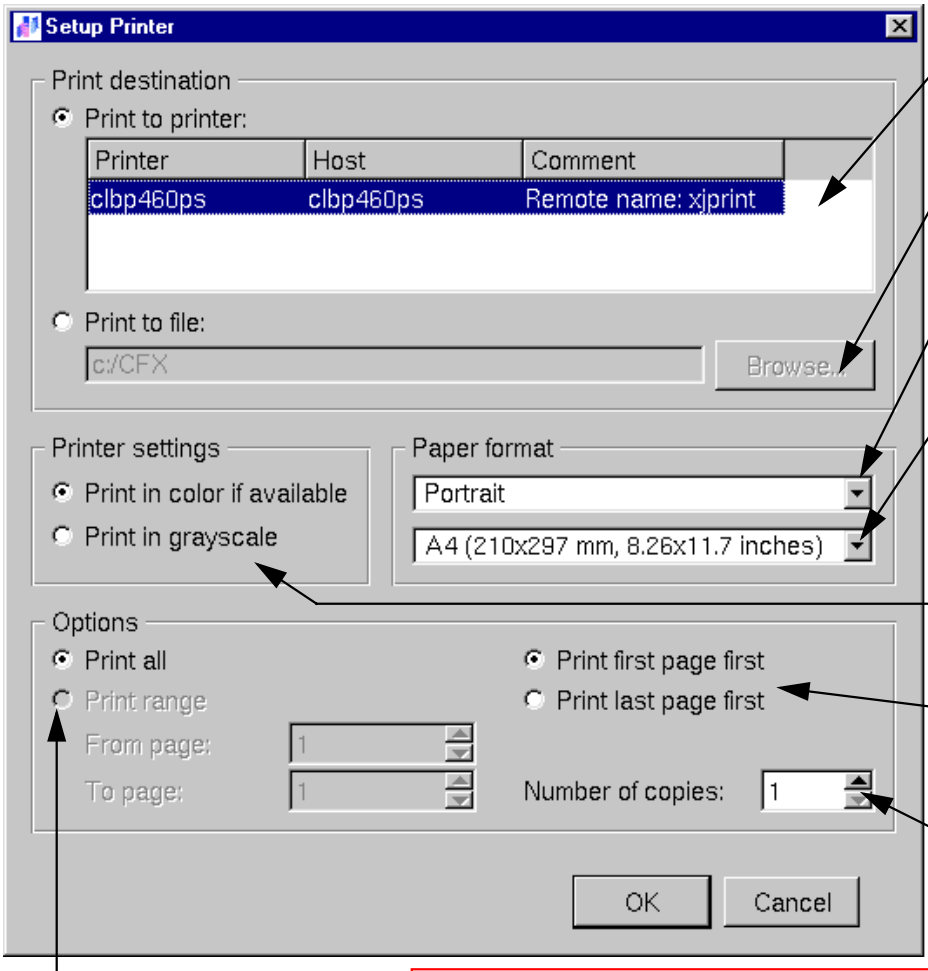
More Help:

- **Saving an Image (p. 20 in CFX-5 Solver and Solver Manager)**
- **Printing and Saving Images (p. 20 in CFX-5 Solver and Solver Manager)**
- **The Graphics Window. (p. 5 in CFX-5 Solver and Solver Manager)**

Choose the type of image from the available options, enter a filename for the image and click **Save**.

Printing directly to a printer

To print the image directly to a printer, or to create a printable postscript image, right-click in the graphics window and select **Print**. On UNIX systems the form shown below appears. On Windows systems the generic Window print panel appears.



If printing to a printer, select the printer from the list.

If saving a printable image to file, click here to browse to the save location

Select between Portrait and Landscape image orientation.

Choose a paper size from **A0** to **A9**..

Choose whether to print in colour or greyscale (if applicable to your printer).

You can reverse the print order by selecting **Print last page first**.

Select the number of copies to be printed.

Choose whether to print everything or specify a range.

More Help:

- **Printing directly to a printer (p. 21 in CFX-5 Solver and Solver Manager)**
- **Printing to a file (p. 22 in CFX-5 Solver and Solver Manager)**
- **Printing and Saving Images (p. 20 in CFX-5 Solver and Solver Manager)**

If printing to a printer, select the printer name from the list. You can choose a Paper format of Portrait or Landscape and a paper size between A0 and A9. If your printer can print in colour you can make use of the **Print in color if available** function. If you select this option but your printer cannot print in colour, it will produce the same image as if you had selected **Print in greyscale**.

You may also select the number of copies and the pages to print in the **Options** section of the form.

Click **OK** to print.

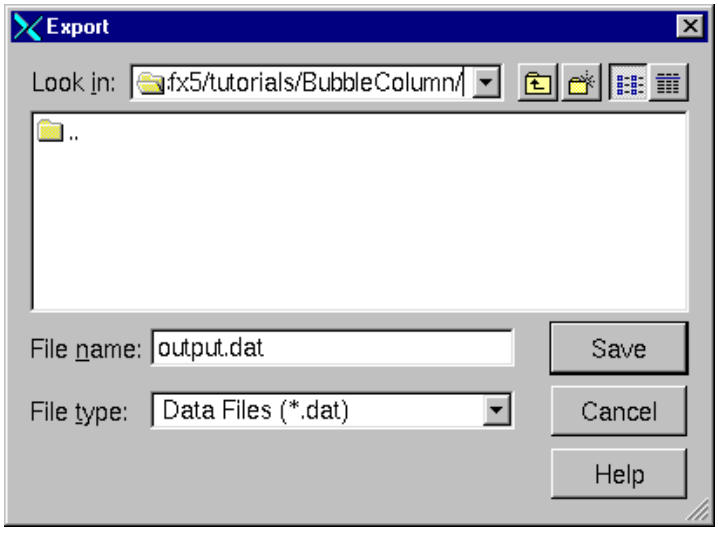
Printing to a file

To print a postscript image to a file, simply select **Print to file** and click on the **Browse..** button. Enter the filename and click **Save**. Then click OK on the Setup Printer form.

Note: Clicking Save in the above instruction does not save the image. You must click **OK** to save the image.

Exporting Plot Data

You can export the data from any plot monitor by right-clicking in the window of your choice and selecting Export Plot Data. The following window will appear for you to enter a filename for the saved data.



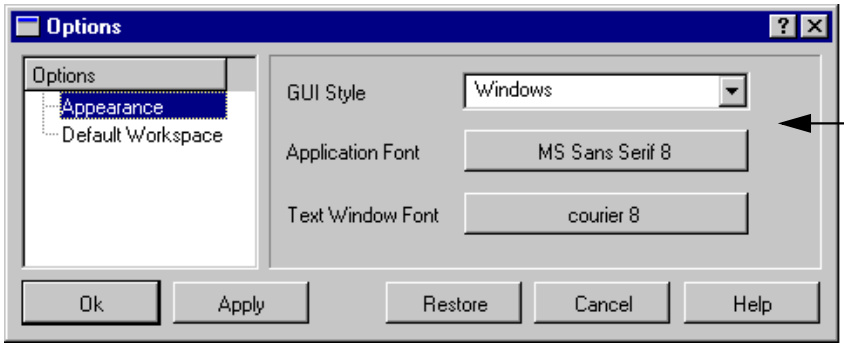
Exported data is saved as .dat files. The data format consists of comma-delimited entries that can be imported into e.g. spreadsheet programs or other post-processors.

The data format for the exported file consists of comma-delimited entries. You can then use the data as the basis for import to another application. All files are saved with a `.dat` extension by default.

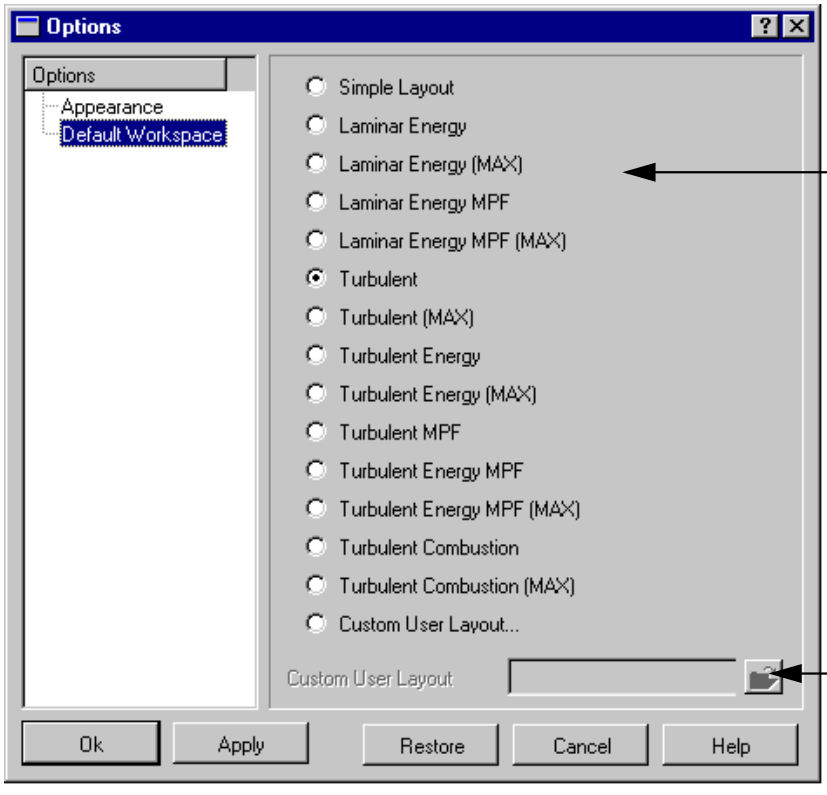
Edit

The Edit menu allows you to set options for the CFX-5 Solver Manager. The Options window allows you to set the following:

Options



Choose a **GUI Style** from the available option. You may also wish to change the main **Application Font** and the font that appears in the **Text Window**.



There are a number of default options available which organise the workspace according to the type of simulation you are carrying out. For example, for studying a turbulent flow without heat transfer, simply select **Turbulent** (as above). If the run includes heat transfer, selecting **Turbulent Energy** will create an extra graphics window showing the residuals associated with Thermal Energy. It is also possible to create and load your own workspace templates (please see **Creating a Custom User Layout (p. 36 in CFX-5 Solver and Solver Manager)**).

To load a a custom designed layout, click here and then on the open file symbol to load the layout.

More Help:

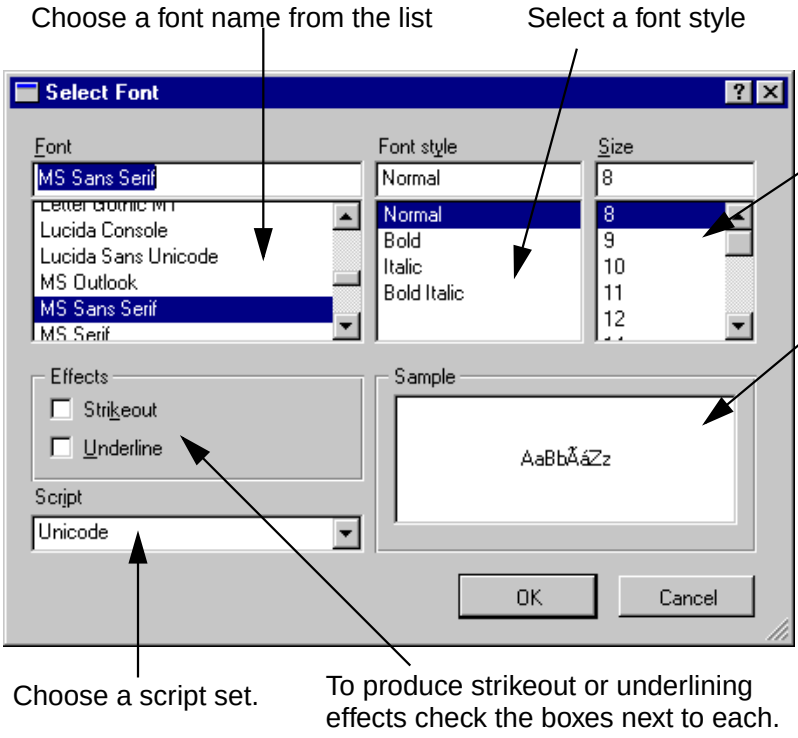
- **Edit (p. 23 in CFX-5 Solver and Solver Manager)**
- **Customising the Solver Manager (p. 9 in CFX-5 Solver and Solver Manager)**
- **The CFX-5 Solver Manager Main Menu Bar (p. 11 in CFX-5 Solver and Solver Manager)**

GUI Style

There are a number of GUI styles that can be chosen from, and are available on all platforms. For example, choosing a Windows style will change the look and feel of the GUI to resemble that of a Windows application. You can select from Windows, Motif, Motif Plus, SGI, Platinum and CDE (Solaris) styles. Once you have selected a style click **Apply** to test.

Application Font and Text Window Font

To change the fonts that will appear anywhere outside of the main text output window. The Text Window Font applies only in the text window where the text information for a run is displayed. Clicking on the button that shows the current font will open the Select Font panel.

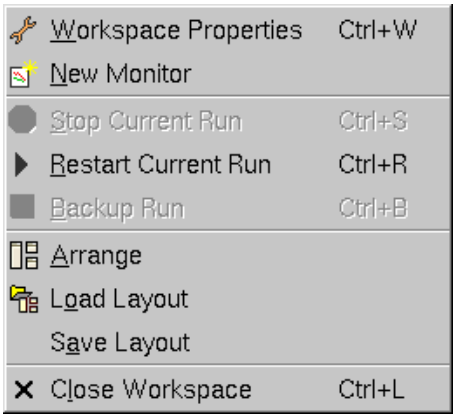


More Help:

- **Application Font and Text Window Font (p. 25 in CFX-5 Solver and Solver Manager)**
- **Edit (p. 23 in CFX-5 Solver and Solver Manager)**
- **GUI Style (p. 25 in CFX-5 Solver and Solver Manager)**

Workspace

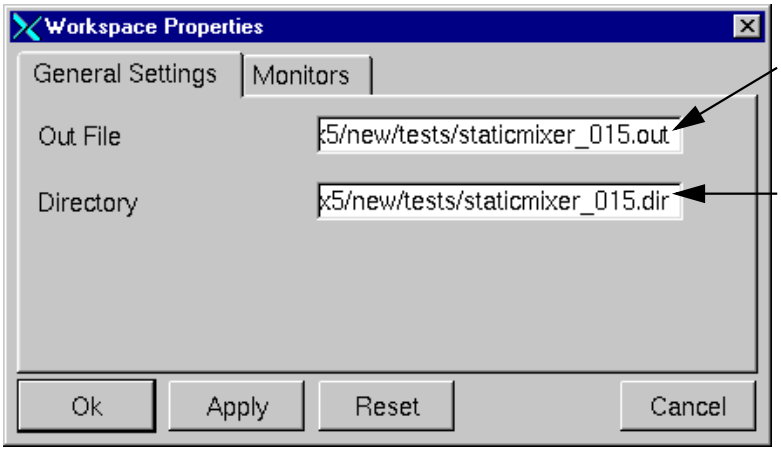
The Workspace menu controls plots and text windows which are visible in the viewer. The current run can also be backed up, restarted or stopped. The following options are available:



Workspace Properties

General Settings

When editing a text monitor, the location of the .out file and the .dir directory created by the solver are given. These boxes may not be edited and are for information about the workspace.



The **Out File** box shows the location and name of the output file from the solver.

The **Directory** box shows the name and location of the directory created

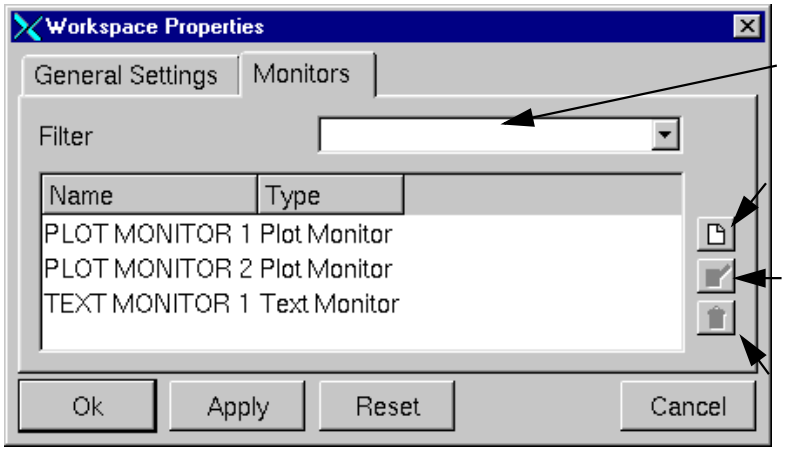
It should be noted that the boxes themselves are not editable and are there for information purposes only. The location of the Out File and Directory can be set using the **Working Folder** option when defining the run (see **Define Run (p. 11 in CFX-5 Solver and Solver Manager)** for more details).

☞ More Help:

- **General Settings (p. 26 in CFX-5 Solver and Solver Manager)**
- **Workspace (p. 26 in CFX-5 Solver and Solver Manager)**

Monitors

The Monitors tab menu allows for customisation of the Solver Manager viewer and text windows.



The Filter option can be activated to show only Plot Monitors, Text Monitors or both.

Click to create a new monitor.

When one of the monitors in the list is highlighted, clicking here will allow you to change its properties.

When one of the monitors in the list is highlighted, clicking here will delete it.

When creating or editing a monitor, new forms will appear. Please see **Monitors: General Settings (p. 29 in CFX-5 Solver and Solver Manager)** and **(p. 32 in CFX-5 Solver and Solver Manager)**.

☞ More Help:

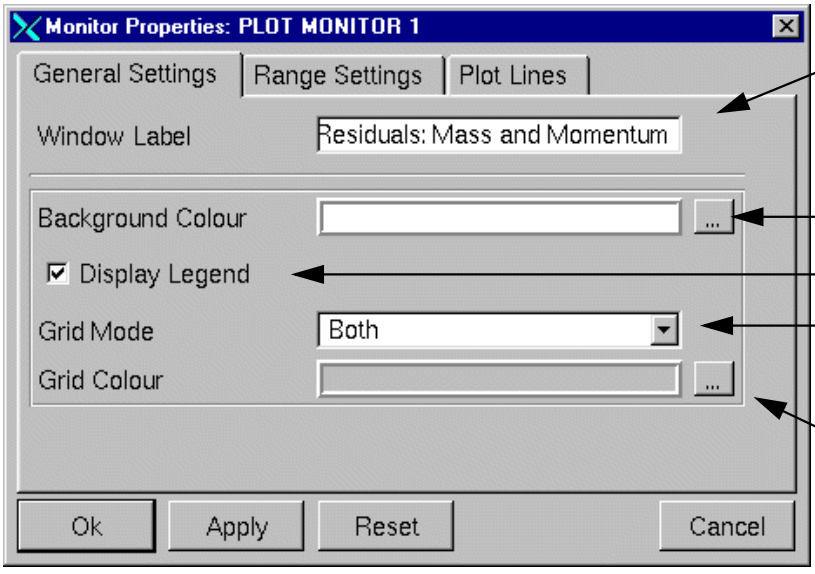
- **(p. 32 in CFX-5 Solver and Solver Manager)**
- **Workspace (p. 26 in CFX-5 Solver and Solver Manager)**
- **The CFX-5 Solver Manager Main Menu Bar (p. 11 in CFX-5 Solver and Solver Manager)**

To create a new monitor, click on the icon. You will be asked to enter the name and type (either **PLOT MONITOR** or **TEXT MONITOR**) of monitor you wish to create.

After you have created a new plot (by clicking on the icon in the **Monitors** tab menu), or have clicked the edit icon to change the properties of an existing plot, the **Monitor Settings** form will appear with 2 tab menus available, **General Settings** and **Plot Lines**. See **Monitors: General Settings (p. 29)** and **(p. 32)** for further information.

To delete a plot, highlight it and click on the icon.

Monitors: General Settings



Displays the current plot window title. Enter a new string here to change.

Click to change the background colour

Toggles visibility of the legend

Select if grid lines are to be drawn on X, Y, both or no axes.

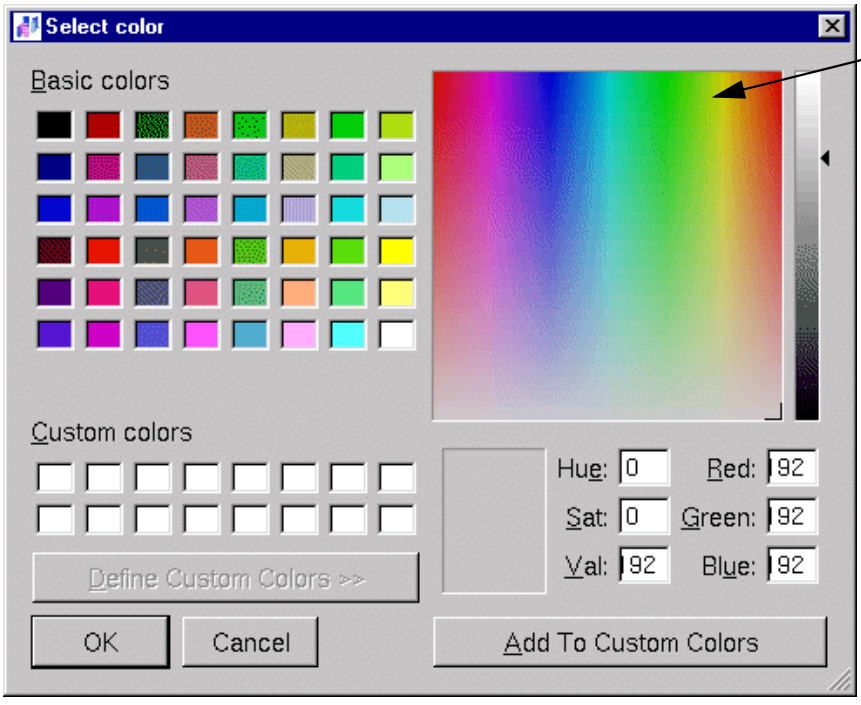
Select a grid colour

☞ **More Help:**

- **Monitors: General Settings (p. 29 in CFX-5 Solver and Solver Manager)**
- **The Graphics Window. (p. 5 in CFX-5 Solver and Solver Manager)**
- **Workspace (p. 26 in CFX-5 Solver and Solver Manager)**

You can change the name of the **Window Label** by entering a new name in the box.

You can select a new **Background Colour** by clicking on the ... icon. A different **Grid Colour** can be chosen in the same way.



Select a Colour from **Basic Colours** or **Custom Colours**. To create a Custom Colour, use this window to select a colour and click on **Add To Custom Colours**. Alternatively enter values for Red, Green and Blue.

More Help:

- General Settings (p. 26 in CFX-5 Solver and Solver Manager)
- Workspace Properties (p. 26 in CFX-5 Solver and Solver Manager)
- Workspace (p. 26 in CFX-5 Solver and Solver Manager)

To toggle visibility of the Legend, simply check the **Display Legend** box ON or OFF. To plot the values on the Y axis using a log scale, ensure that the **Log-Scale Y** box is checked.

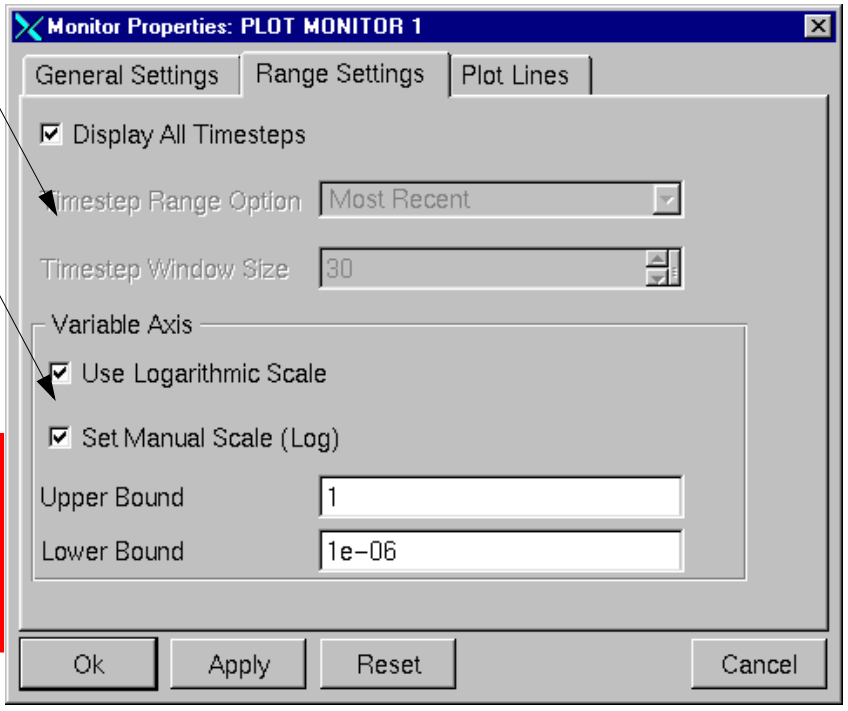
The **Grid Mode** selector allows you to choose which grid lines are visible in the graphics window. The options available are **Both**, **X**, **Y** and **None**.

To change the colour of the grid lines, click on the ... icon and select a colour.

Monitors: Range Settings

You can choose to display All Timesteps, the Most Recent (enter a value for the number to be displayed), a Fixed region (enter the start and end number) or iterations for This

Choose whether to display a Log scale (in which case you can set a lower and upper bound), or a real number scale.



More Help:
- The Graphics Window. (p. 5 in CFX-5 Solver and Solver Manager)
- Workspace (p. 26 in CFX-5 Solver and Solver Manager)

You can specify how much of the plot you wish to view using the range controls.

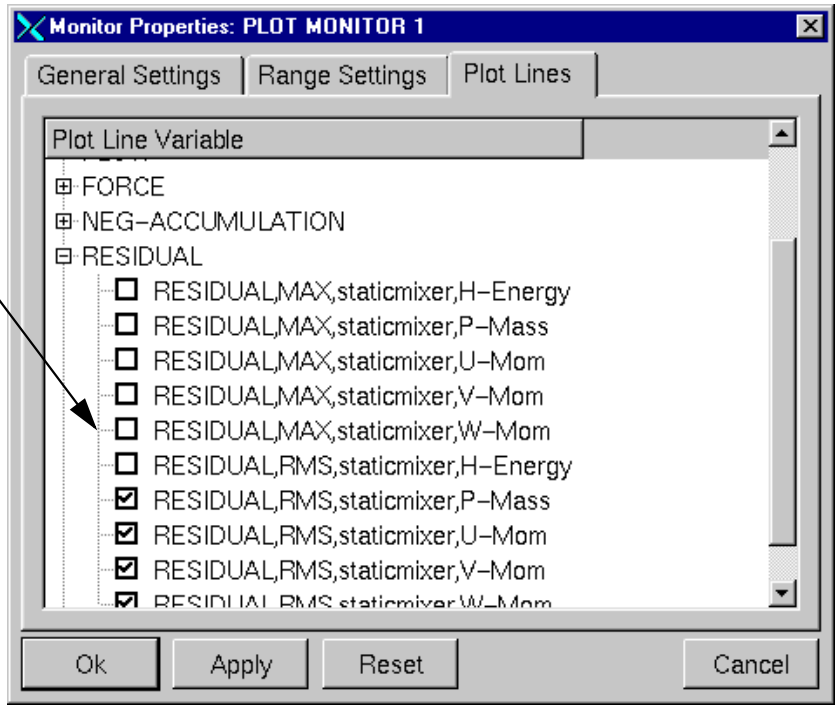
- **Display All Timesteps** will show values for every iteration. If you are viewing a restarted run, results from the previous run will also be visible using this feature.
- **Most Recent** will display the current iteration and a number of previous iterations, set by entering a value into the Timestep Window Size box.
- **Fixed** allows you to enter a beginning and end iteration which will always be displayed, regardless of the current iteration number.
- **This Run Only** will display the range for the current run. If the run is a restart, any previous runs will not be included in the range.

You can select whether to view a **Logarithmic** scale or a real scale. When viewing a log scale, you can enter an **Upper** and **Lower Bound** for the variable axis.

Monitors: Plot Lines

To expand or reduce a list for a given category, click on the + or - sign next to it.

Visibility for each variable is toggled by checking the box next to the variable



More Help:

- (p. 32 in CFX-5 Solver and Solver Manager)
- The Graphics Window. (p. 5 in CFX-5 Solver and Solver Manager)
- Workspace (p. 26 in CFX-5 Solver

To view the available plots for a particular category, click on the **+** sign next to it. In the diagram above, the options for BALANCE and RESIDUAL have been opened. To make a plot visible, check the box next to the plot. In the diagram above, all of the BALANCE and RESIDUAL plots were checked to make them visible in the graphics window.

A brief description of each of the variable types is given below:

RESIDUAL

A description is given in **Residual plotting (p. 65)**

FORCE

A description is given in **Calculated Wall Forces and Moments (p. 117)**.

NEG ACCUMULATION

Negative accumulation is the transient term contribution to the balance equation. See the information under BALANCES for further details.

FLOW

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

BALANCES

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

SOURCES

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 setup, or the entire domain. SOURCE values which are defined for the entire domain are automatically computed by the solver, and vary depending on the models which are being used. For example, automatically computed SOURCE values will appear in the turbulent kinetic energy equation which represent the production and dissipation of turbulence.

TIMESTEP

The timestep values are plotted.

ACCUMULATED TIMESTEP

The accumulated timestep values are plotted.

USER

If you have created any monitor points (see **Monitor Points (p. 87)**), a USER category will be available. You can expand this category to select the monitor points to plot.

To commit the new settings, click on **Apply**.

Further details on the output of the solver can be found in **The CFX-5 Output File (p. 106)**.

Stop Current Run

Selecting **Stop Current Run** or clicking the  icon instructs the CFX-5 Solver to stop at the end of the next iteration or timestep. It is only available when the **Current Run** is in the **Running** state. When the CFX-5 Solver actually stops, the run will be labelled as **Finished**, and the message “<run> has terminated at user’s request. Results are in <directory>. Run concluded at <time>” will be displayed. <run> will be the name of the results file without the .res extension, <directory> will be the working directory you are in (or have specified), and <time> will be the time at which the run stopped.

You may now do anything to a manually terminated run that you could do to a fully completed run.

If you run the CFX-5 Solver from the command line, you can imitate the **Stop** button by using the command `cfx5stop`. See **Starting the CFX-5 Solver from the Command Line (p. 68)** for more details.

Restart Current run

This option, also available when you click on the  icon, allows you to restart a run that has finished. There are many different ways in which you can set up a restarted run, please see **Restarting a Run (p. 55)** for full details. Issuing a "Restart" on a run will start the run with the same settings as the previous run, including Parallel settings.

Backup

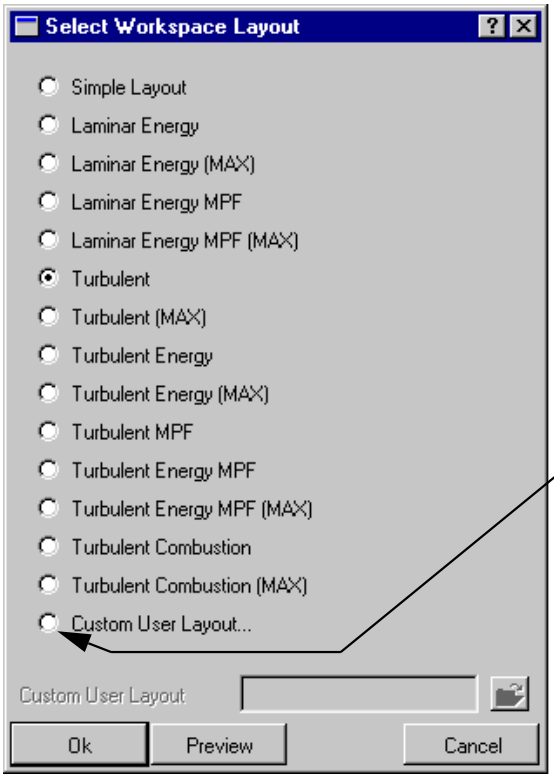
Clicking on the  icon instructs the CFX-5 Solver to write a backup file of the results so far, at the end of the timestep that it is currently calculating. This file contains sufficient information to use for restarting or for visualisation. It can be found in a sub-directory with the same name as the run, in the working directory.

Arrange

Arrange, which can also be carried out by clicking on the icon, will organise all the windows in the workspace with no overlap. The mass and momentum window will appear first, to the left (and top if there are more than 3 windows) and the text window(s) will appear on the right (and at the bottom if more than 2 windows exist). All other windows will appear in between.

Load Layout

This form is the same as the one available for **Default Workspace**, which can be found in the **Options** section of the **Edit** menu (**Edit (p. 23 in CFX-5 Solver and Solver Manager)**.



There are a number of default options available which organise the workspace according to the type of simulation you are carrying out. For example, for studying a turbulent flow without heat transfer, simply select **Turbulent** . If the run includes heat transfer, selecting **Turbulent Energy** will create an extra graphics window showing the residuals associated with Thermal Energy. It is also possible to create and load your own workspace templates, please see **Creating a Custom User Layout (p. 36 in CFX-5 Solver and Solver Manager)** for more details.

To load a a custom designed layout, click here and then on the open file symbol to load the layout.

More Help:

- **Workspace (p. 26 in CFX-5 Solver and Solver Manager)**
- **Customising the Solver Manager (p. 9 in CFX-5 Solver and Solver Manager)**
- **The CFX-5 Solver Manager Main Menu Bar (p. 11 in CFX-5 Solver and Solver Manager)**

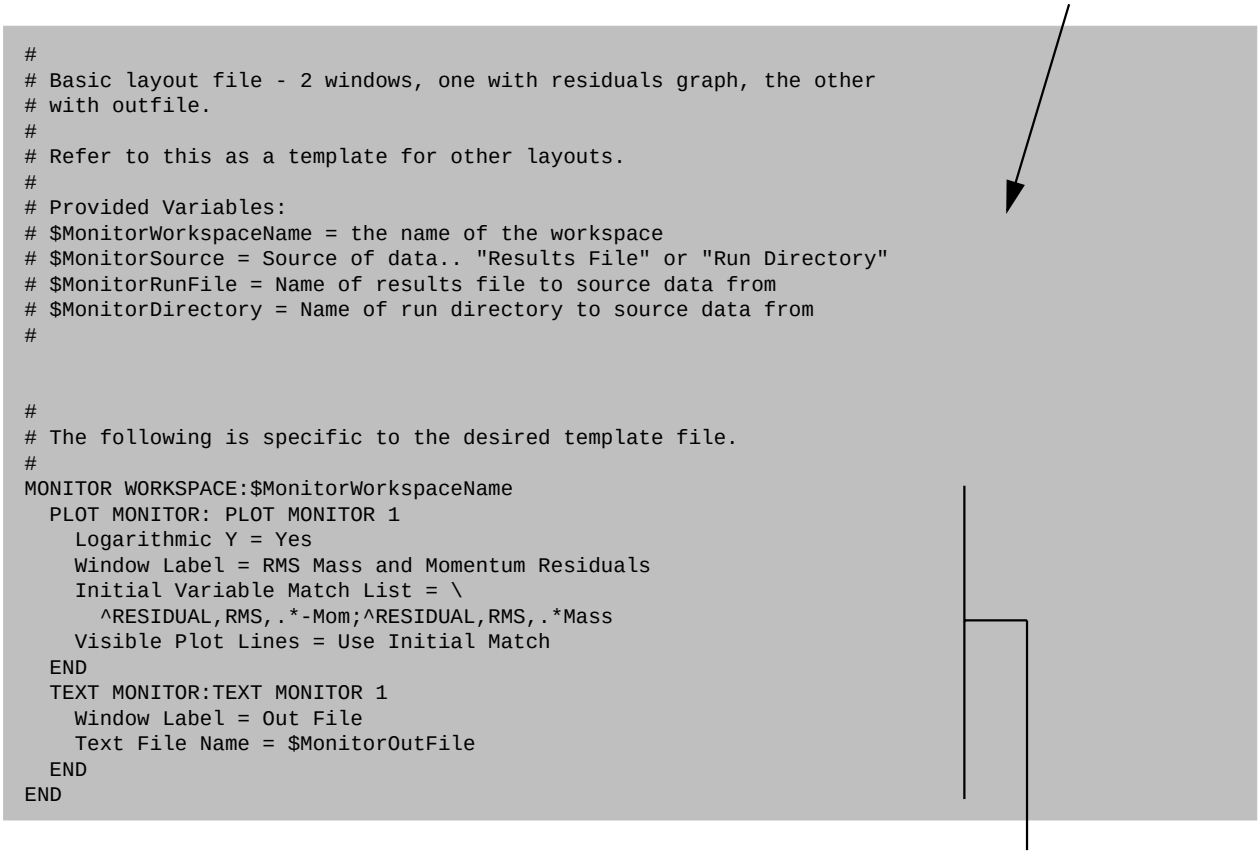
Creating a Custom User Layout

As a result of the large number and diverse range of CFD simulations that can be solved in CFX-5, it may be necessary to create a custom layout with which to monitor the progress of your run. You may wish to edit the layout using the Workspace Properties function. As it is not possible to save an edited layout in the GUI however, you may wish to create a `.mst` file which can be reloaded at any future time. The following section explains the format of a `.mst` file and provides instructions for editing or creating your own.

Structure of the `.mst` file

The following example was taken from the default **Simple Layout** available from the **Select Workspace Layout** form (found in **<CFXROOT>/etc/5.5.1/)**.

Explanatory comments



```
#
# Basic layout file - 2 windows, one with residuals graph, the other
# with outfile.
#
# Refer to this as a template for other layouts.
#
# Provided Variables:
# $MonitorWorkspaceName = the name of the workspace
# $MonitorSource = Source of data.. "Results File" or "Run Directory"
# $MonitorRunFile = Name of results file to source data from
# $MonitorDirectory = Name of run directory to source data from
#


#
# The following is specific to the desired template file.
#
MONITOR WORKSPACE:$MonitorWorkspaceName
  PLOT MONITOR: PLOT MONITOR 1
    Logarithmic Y = Yes
    Window Label = RMS Mass and Momentum Residuals
    Initial Variable Match List = \
      ^RESIDUAL,RMS,.*-Mom;^RESIDUAL,RMS,.*Mass
    Visible Plot Lines = Use Initial Match
  END
  TEXT MONITOR:TEXT MONITOR 1
    Window Label = Out File
    Text File Name = $MonitorOutFile
  END
END
```

This section tells the Solver Manager how many windows to create and which variables to plot. First, the name of the plot monitor and the window label are set. Both can be changed by editing the text on the right hand side of the ":" or "=". Semi-colons (**;**) **MUST** be used to separate variables.

For a PLOT MONITOR, the Initial Variable Match List gives a list of variables to search for and plot.

The TEXT MONITOR searches for the file name stored in $MonitorOutFile. There should be no need to change settings for the text monitor.

For each plot monitor, the variables to be plotted are entered into the **Initial Variable Match List** and searched for. If found, they are plotted in the graphics window and added to the legend with an assigned colour. The variable to be searched for must be entered in the same way that variables appear in the Plot Lines tab menu of the Monitors form (please see **Monitors: Plot Lines (p. 32)** for more information).

The variable name should be preceded by **^** , which stipulates that the next character must be at the start of the search string. For example, if “a” was entered as the search string, a match could be found for “aaa” and “data”. If the search string “^a” was entered, only “aaa” could be returned.

All variables must be separated by a semi-colon (**;**).

To explain the search string, the following line shows the full variable name for the RMS residual of U Momentum in a domain called “Combustor”.

```
RESIDUAL,RMS,Combustor,U-Mom;
```

In the **example**, the following line was entered:

```
^RESIDUAL,RMS,.*-Mom
```

The **.*** is entered to allow any character string between **RESIDUAL,RMS,** and **-Mom** to be included. The **.** describes any one character and the ***** acts on the **.** so that **.*** collectively describes any number of characters. The **** acts as an escape character.

For the case of the **example**, we can expect that all three components of momentum and mass will be plotted. Each component of momentum or mass could also have been specified separately as follows:

```
^RESIDUAL,RMS,.*U-Mom;^RESIDUAL,RMS,.*V-Mom;\
^RESIDUAL,RMS,.*W-Mom;
```

Plotting your own variables

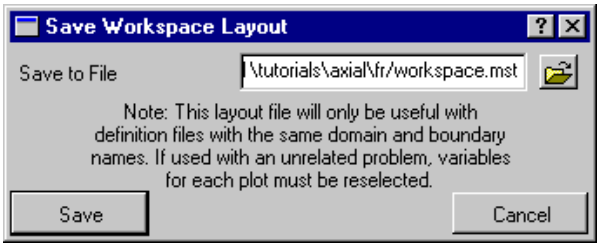
The easiest way to create your own plot monitor is to copy the definition of an existing monitor and edit it to include the new variables. The default monitor files can be found in **<CFXROOT>/etc/5.5.1/** directory. For example, if you wish to view turbulence data on the same plot as mass and momentum you could enter the following:

```
^RESIDUAL,RMS,.*-Mom;^RESIDUAL,RMS,.*-Diss.K; \
^RESIDUAL,RMS,.*-TurbKE;
```

As stated previously, the string to search for can be found from the variable lists in the Plot Lines tab menu of the Monitors form (please see **Monitors: Plot Lines (p. 32)** for more information)

Save Layout

Currently this option is only available for problems with the same domain and boundary names. It will save the current workspace layout to a file which can be reloaded at another time.



This feature allows you to save the current layout to file. As the warning message indicates, this feature may only be used for definition files with the same domain and boundary names. Generic templates can be created; please see **Creating a Custom User Layout (p. 36 in CFX-5 Solver and Solver Manager)** for more details.

More Help:

- **Save Layout (p. 39 in CFX-5 Solver and Solver Manager)**
- **Creating a Custom User Layout (p. 36 in CFX-5 Solver and Solver Manager)**
- **Workspace (p. 26 in CFX-5 Solver and Solver Manager)**

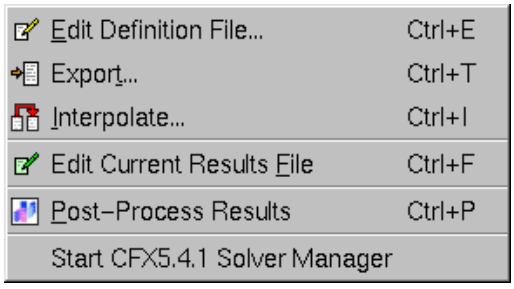
This feature is useful when carrying out different runs on the same problem, for example changing a boundary condition value or turbulence model. If you wish to create a layout that can be used on other problems, you should follow the instructions given in **Creating a Custom User Layout (p. 36 in CFX-5 Solver and Solver Manager)**. If you load other results into the CFX-5 Solver Manager using a layout saved using Save Layout, you will have to reselect the variables for each plots.

Close Workspace

Closing the workspace will close all windows related to the current run. Any other runs that were open will not be affected, and the last run that was open before the current run will return to view. If the solver was in progress on the current run it will continue to operate in the background, and the run can be re-monitored by using the **Monitor Run in Progress** feature in the **File** Menu (**Monitor Run in Progress (p. 19)**).

Tools

The **Tools** menu has the following options:



A summary of what each option will do is given below. This is followed by a detailed description of each one.

Edit Definition File... The **Edit Definition File** option is used to make minor changes to the command file section of a definition file. Upon selecting this option, a second window will appear allowing you to choose any existing definition or results file. This will be followed by the **Definition File Editor**. See **Editing a Definition File (p. 75 in CFX-5 Solver and Solver Manager)**.

Export... This option is used to produce files for use with post-processors other than CFX-Post or CFX-Visualise. See **Export (p. 41)**.

Interpolate This option allows you to impose a solution from a results file containing one grid, into a definition file containing a different grid. See **Interpolate (p. 41)** for more information.

Edit Current Results File Edit the results file in the current workspace (if available). See **Edit Current Results File (p. 45)**

Post-Process Results This option loads the CFX-Post post-processor. See **Post-Process Results (p. 45)**.

Start CFX-5.4.1 Solver Manager This feature can be utilised when you wish to view results files written by previous versions of CFX-5. See **Start CFX-5.4.1 Solver Manager (p. 45)**

Edit Definition File

The **Edit Definition File** option can be used to make minor changes to a definition file, without revisiting CFX-Build. This is particularly useful when you wish to re-run a simulation with only slightly different model parameters.

For a complete description of the Definition File Editor, see **Editing a Definition File (p. 75)**.

Export

Selecting **Tools>Export** brings up the **Export** form, which allows you to export the data in a Results File in a suitable format for input into post-processors other than CFX-Post. You can also run the CFX-5 Export utility from the command line.

The Export utility is documented in **The File Export Utility (p. 139)**.

Interpolate

Interpolation allows you to transfer a solution from a results file, containing one grid, onto a second file containing a different grid.

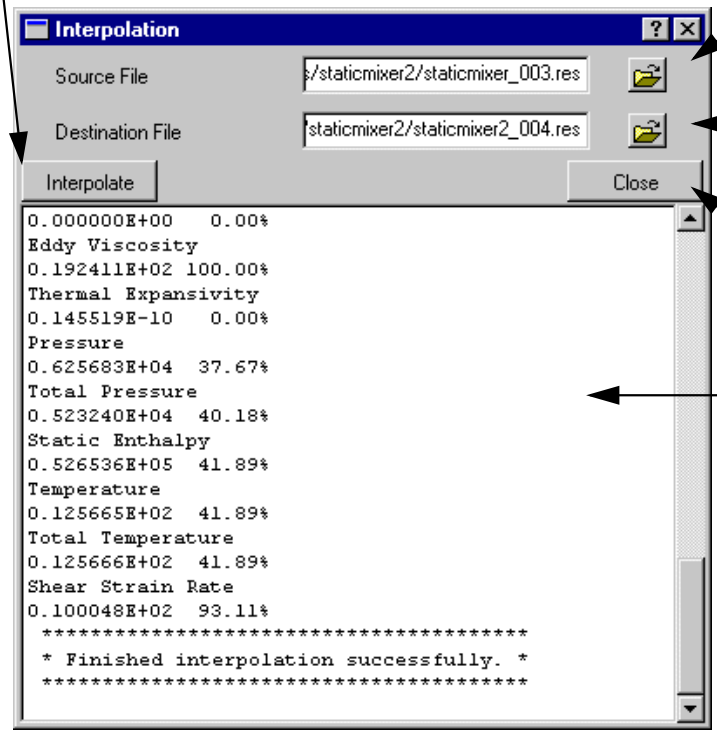
The major benefit of interpolation is the ability to use the solution from a simple model as a good initial guess of a more complex model. By using interpolation, you are increasing the likelihood of converging a complex model.

You can also interpolate to slightly modified geometry or boundary conditions etc. to facilitate rapid parametric design studies.

Using the CFX-5 Solver Manager to Interpolate Results

The easiest way to set up an interpolation run is to use the CFX-5 Solver Manager Tools menu. When you select **Tools>Interpolation**, the following form appears:

Click to begin interpolation process.



Select the source (**.res**) file.

Select the destination file (which can be either a **.res** or **.def** file.

Click here to close the form. If interpolation is in progress, it will continue in the background.

The output window displays details of the interpolation process.

More Help:

- **Interpolate (p. 41 in CFX-5 Solver and Solver Manager)**
- **Tools (p. 40 in CFX-5 Solver and Solver Manager)**

Using this form, select the file that you'd like to interpolate the solution from, as well as the file that you'd like the solution interpolated onto. Use the **Browse** button to change directories and find the files that you want to use. You can interpolate a CFX-5 results (*.res) file onto either an existing CFX-5 definition (*.def) file, or onto another CFX-5 results (*.res) file.

Important: If you elect to interpolate a CFX-5 Results file onto a second CFX-5 Results file, the interpolator will generate a comparison of the two files. These comparisons will be stored as additional variables with the prefix *Difference* in the file in which the results were interpolated to. For example, *Difference Pressure* is a variable that contains the difference in Pressure between the two files. These variables can be used in CFX-Post to determine regions where the solution has changed significantly.

When you have selected the files, press **Start** to begin the interpolation. The Start button calls the `cfx5interp` script which performs the file interpolation. The output window prints the details of the interpolation process. If you wish to save the contents of the output to a text file, right-click in the window and select **Save As**. Enter the name of the text file and click **Save** to save. You can also search for any words within the text field by right-clicking in the window and selecting **Find**.

The `cfx5interp` script uses conservative values when interpolating results.

There may be situations where the shape of the model has changed and the solution set no longer maps exactly onto the model grid. If the solution grid does not provide values for all of the nodes in the new grid, then the `cfx5interp` script will extrapolate values for the extraneous points, based on the given solution set.

Using the Command Line to Interpolate Results

In some instances, you may wish to use the `cfx5interp` script directly from the command line to initiate an interpolation.

To use the `cfx5interp` script, enter the following in the command line:

```
cfx5interp -def <interpolate to file> -res
<interpolate from file>
```

This command will produce the same results as having run the **Tools>Interpolation** function from the CFX-5 Solver Manager.

The `cfx5interp` script is also capable of producing a text file of results for specific locations within the fluid domain. This is particularly useful if you have experimental data that you wish to validate.

To produce a results text file, you must first create a text file containing the particular coordinates of interest. The vertex text file should be of 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, you can run the following `cfx5interp` script:

`cfx5interp -vtx <vertex file> -res <results file>`

The script will create a third file entitled `<vertex file>.new`. This new text file will contain the coordinates specified in the vertex file, as well as the results at these locations.

If there exists a set of coordinates that lie outside of the solution grid, values of 0.0E0 will be interpolated for all variables at this location. The cfx5interp script will not extrapolate results to a vertex file.

A sample file, with variable descriptions, is shown below:

```
1 : Coordinates + variables for vertex
 0.000000E+00  0.000000E+00 -2.000000E+00                    Vertex coordinates
    3 : G/PRESGRD   1.552757E+02 -1.102330E+02 -1.906295E+03 Pressure (x, y,z)
    1 : G/DENSITY   1.000000E+03                             Density
    1 : G/CONDUCT   6.200000E-01                             Conductivity
    1 : G/VISCDYN   1.140000E-03                             Dynamic Viscosity
    3 : G/VEL  -9.742797E-03 3.445983E-03 -3.966365E+00      Velocity
    1 : G/SPHEATP   4.190000E+03                             Specific Heat
    1 : G/ENTH   1.256984E+06                                Enthalpy
    1 : G/TEMP   2.999962E+02                                Temperature
    1 : G/PRES   4.147881E+01                                Pressure
    1 : G/TEMPTOT   2.999981E+02                             Total Temperature
    1 : G/PRESTOT   7.966394E+03                             Total Pressure
    0 : END OF VARIABLES
```

Note: Some of the values obtained using the cfx5interp script may differ slightly from the values obtained using Data Export in CFX-Post. These minor discrepancies result from different methods of calculation. Discrepancies are more likely to occur at points which lie very close to the edge of the mesh elements or in regions of prism and hexahedral elements. Inconsistencies are likely to be more significant where gradients are large, particularly in the boundary layer.

Edit Current Results File

This feature is available when you have a results file in the current workspace that has finished. Selecting **Tools>Edit this Results File** will start the Definition File Editor and automatically open the current results file for editing. For further details please see **Editing a Definition File (p. 75)**.

Post-Process Results

Selecting **Post-Process Results** or clicking on the  icon starts up CFX-Post. If a finished run is currently available you will be asked if you want to load CFX-Post with the results file, or without. If you choose to load without loading the current results file, you may still load it using CFX-Post. The CFX-5 Solver Manager remains open while CFX-Post is running.

You may shut down the CFX-5 Solver Manager (using **File>Quit**) once CFX-Post has started.

More details on CFX-Post can be found in **Overview of CFX-Post (p. 1 in CFX-Post)**.

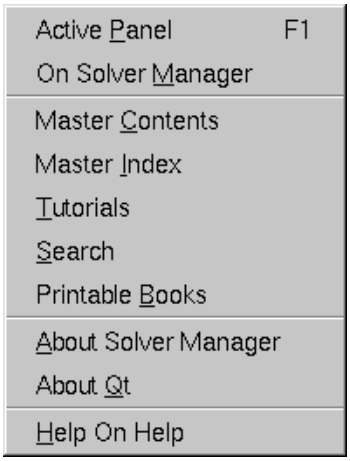
Start CFX-5.4.1 Solver Manager

The previous version of the CFX-5 Solver Manager can be used if you wish to view results files that were created in previous versions of CFX-5. The current version of the CFX-5 Solver Manager is unable to read older **.res** files.

A message will appear asking for confirmation. Click **Run** to open the older Solver Manager.

Help

The Help menu contains options for navigation to sections of the online documentation.



Active Panel F1

This feature will open context-sensitive help for the panel currently highlighted. For example, if you currently have the Viewer selected as the active window, selecting **Active Panel** (or pressing the **F1** key) will open a help window with information about the Viewer.

On Solver Manager

The help section for the CFX-5 Solver Manager will be invoked.

Master Contents

This will take you to the main CFX-5 help page, which gives links by component and section.

Master Index

The CFX-5 index contains entries for all of the documentation.

Tutorials

Overview and links to the CFX-5 Tutorials.

Search

This opens a help document explaining how to use the powerful search feature available in Adobe Acrobat Reader.

Printable Books

Many of the helpfiles are grouped into larger sections and presented in PDF format for easy printing.

About

Lists information about the selected product.

Help on Help

Opens a document explaining how to navigate through on-line help in CFX-5 using different methods.

How to use the CFX-5 Solver Manager

The following section describes procedures for specific types of run that you may wish to carry out. Often, it will reference the documentation in the previous section as all forms and menu items have been described already.

What you need to do depends on whether you need an Initial Values file or not, and whether the Definition File name has already been passed to the CFX-5 Solver Manager.

When do you need to specify an Initial Values file?

You need to specify an Initial Values file if you selected not to write a mesh to your Definition File.

> If you have already generated a mesh for your problem and it is contained within a CFX-5 Results File, then you can choose in CFX-Build to use that mesh instead of re-writing the mesh when you generate your Definition File. This is accomplished by turning **Off** the **Write Mesh** toggle on the **Definition File** form in CFX-Build.

You may also wish to specify an Initial Values file if you have written a mesh in your Definition File. If you choose to do so, then the CFX-5 Solver will:

- use the mesh from the Initial Values file
- read Initial Values from the Initial Values file for all variables that have Initial Conditions set to Automatic or Automatic with Value in the specified Definition file
- use CFX-5 Solver default or user-specified values for all variables that have Initial Conditions set to Default or Value in the specified Definition file.

For more information on setting Initial Values, see **Setting the Initial Conditions (p. 1179 in CFX-Build Chapter 8)**.

When is the name of your Definition File passed to the CFX-5 Solver Manager?

The name of your Definition File is automatically passed to the CFX-5 Solver Manager if you started it by any of the following methods:

1. by selecting **Start** CFX-5 Solver Manager from the **Write Definition File** form in CFX-Build.

2. by starting it from the command line using the `-def` option. See **Starting the CFX-5 Solver Manager from the command line (p. 4)** for more details.

Using the CFX-5 Solver Manager for a Serial Run

A serial run is the default way of running a CFD simulation. During a serial run, all of the computation will be done by a single process running on one machine. You can also make CFX-5 run your problem in parallel, where the computation is divided into more than one process and is done on more than one machine. If you want to run your job in parallel, you should read and understand the instructions below, and then refer to the section **Setting Up and Running a Parallel Run (p. 52)**.

1. On opening, the CFX-5 Solver Manager will look like the diagram shown in **Solver Manager Objects (p. 5)**. If your definition file was passed to the solver from CFX-Build, the Define Run form will automatically select the correct definition file. You may still need to specify an Initial Values file (see **When do you need to specify an Initial Values file? (p. 48 in CFX-5 Solver and Solver Manager)** for more details) In this case, you can skip steps 2 to 5 and move directly to step 6. Otherwise, you will need to provide the name of the Definition File and/or the Initial Values File and must continue with step 2.

2. You should select the **File**>**Define** option from the main menu. This will open the **Define Run** form (see **Define Run (p. 11)**).

3. The **Definition File** box is used to specify the information that the CFX-5 Solver needs to perform your calculation.

4. In the box labelled **Definition File**, you should specify the Definition File which contains the details of your model, fluid, boundary conditions, mesh and solver parameters, which you created in CFX-Build. If the CFX-5 Solver Manager was automatically passed the name of your Definition File, then this box should already contain the Definition File name and you do not need to change the information.

5. If your Definition File requires you to specify an Initial Values file, then you should enter the name of this file into the box labelled **Initial Values File**. If your Definition File does not require you to specify an Initial Values file, then you will not need to enter anything in this box.

6. Once your Definition File is specified, and Initial Values File if appropriate, you should click on **Start Run** to begin the Solver. Your Run name will be the name of your Definition File with the `.def` removed and a three-digit number added. The number will usually be `001` if this is the first time you have used the CFX-5 Solver Manager for this Definition File. In general, it is the lowest number that would not result in files from previous runs being overwritten. The Run that you have just created will appear in the box at the top of the Main Window. You will be able to see what the CFX-5 Solver is doing from what appears in the Graphics and Text Output areas. First, the axes for a plot appear in the Graphics Area, and a summary of your model properties is printed in the Text Output Area. After a short time, you will start to see tables such as the one shown below, which shows that the CFX-5 Solver has finished initialising your problem and is starting to solve it. Each such table corresponds to one iteration or timestep of your problem (see **Introduction (p. 102)** for what this means). All of the text which appears in the Text Output Area is saved in the CFX-5 Output File, which is described in detail in **The CFX-5 Output File (p. 106)**.

```
================================================================================
OUTER LOOP ITERATION =    1                       CPU SECONDS = 2.68E+00
--------------------------------------------------------------------------------
| Equation | Rate | RMS Res | Max Res | Location |  Linear Solution   |
+----------+------+---------+---------+----------+--------------------+
| U - Mom  | 0.00 | 9.1E-20 | 3.3E-19 |      386 |        0.0E+00  OK |
| V - Mom  | 0.00 | 3.5E+00 | 7.0E+00 |     1418 |        4.0E-03  OK |
| W - Mom  | 0.00 | 1.7E-10 | 6.1E-10 |      386 |        6.8E+07  ok |
| P - Mass | 0.00 | 0.0E+00 | 0.0E+00 |        0 |  8.3  0.0E+00  OK |
+----------+------+---------+---------+----------+--------------------+
```

7. Once results for the second timestep are shown in a table such as the one above, you will start to get data plotted in the Graphics Output Area. The plot for each timestep is the “normalised residual” for each of the equations that the CFX-5 Solver is solving. This is a measure of how inaccurate the current solution is. See **Residual plotting (p. 65)** for more discussion about residuals. These residuals should gradually decrease as the solution progresses. However, don’t worry if they increase on occasional timesteps.

8. When the CFX-5 Solver stops running, you will be given a message in a dialogue box telling you that it has stopped, with some indication of why it has stopped. The CFX-5 Solver may stop:

- if the solution is obtained to the accuracy that you specified
- if it reaches the maximum number of timesteps you specified
- if you stopped it manually
- if your solution is diverging and so caused numerical problems in the CFX-5 Solver

9. These messages (usually with more detail) will also be printed in the Text Output Area, followed by information on the solution itself (if obtained). Often, more information can be found in **The CFX-5 Output File (p. 106)**.

At this point, your Run has finished (or failed). To run the CFX-5 Solver again you would need to define another Run. You can do any of the following:

- define a new Run. Either follow the procedure given above in steps 1 to 5, or see **Define Run (p. 11)** for further details.
- calculate some more timesteps for the original Run. See **Setting Up and Running a Parallel Run (p. 52)**.
- view the results in CFX-Post (provided that the CFX-5 Solver produced a Results File and did not fail). See **Overview of CFX-Post (p. 1 in CFX-Post)** for more details.
- obtain hard copies of the Residual Plots in the Graphics Area. See **Printing and Saving Images (p. 20)**.
- add some comments to the saved version of the text in the Text Output Area. See **Starting the CFX-5 Solver from the Command Line (p. 68)**.
- export the results in the format suitable for input into post-processors other than CFX-Post. See **The File Export Utility (p. 139)**.

You can quit the CFX-5 Solver Manager at any time using the **Quit** button from the **File** menu. This does not stop the CFX-5 Solver calculation, and you can reopen the CFX-5 Solver Manager and take control over the CFX-5 Solver again if you wish.

The CFX-5 Tutorials describe how to use the CFX-5 Solver Manager step-by-step for several different cases, and if you are a new user you are recommended to try at least the first few of these. The Tutorials can be found in **Introductory Tutorials (p. i in CFX-5 Tutorials)** and **Advanced Tutorials (p. i in CFX-5 Tutorials)**.

Setting Up and Running a Parallel Run

You can only run the CFX-5 Solver in parallel if you have purchased an appropriate license. If you are unsure whether you have purchased a license, please refer to your system administrator. Alternatively, read the tutorial **Setting Up to Run in Parallel (p. 217 in CFX-5 Tutorials)** for more information.

For a general description of how the CFX-5 Solver runs in parallel, see **Introduction to CFX-5 Parallel (p. 504)**. You may need to set yourself up to run in parallel; refer to **Parallel Setup for Individual Users (p. 143 in Installing & Introduction to CFX-5)** for details.

To avoid repetition of the documentation, the advanced options available with each type of parallel setup are documented only once, in **Define Run (p. 11)**.

To run the CFX-5 Solver in parallel, there are two major processes which must take place:

1. the partitioning of your mesh into the appropriate number of partitions, and,
2. the running of the CFX-5 Solver on the partitioned problem.

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

Running the partitioning and the CFX-5 Solver as one job

After carrying out steps 1-5 in **How to use the CFX-5 Solver Manager (p. 48)**, continue as described below.

1. Select the **Local Parallel** option in the **Run Mode** box of the **Define Run** panel if you are running the problem with 2 or more processors on the local machine. If you wish to run the problem over 2 or more machines, select **Distributed Parallel**.

2. The Parallel Environment area will expand. The options will be different according to the selection made.

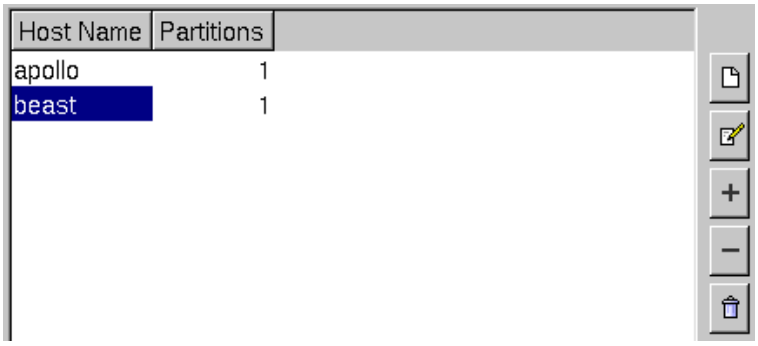
Local Parallel Setup

- Select the number of partitions you require in the **Number of Partitions** type-in box (when you run the job in the CFX-5 Solver, the computation will be divided into this number of processes). You may select between 2 and 254 partitions.
- For Advanced options, check the **Show Advanced Controls** option. When you click on the **Partitioner** tab some new options become available Please see **Partitioner (p. 14)**. The Solver tab will allow you to make further changes. Please see **Solver (p. 17)**.
- Click on **Start Run** to begin the solution process.

Distributed Parallel Setup

- To utilise this feature, a file named **`PVMhosts`** must exist in the `<CFXROOT>/config/<release>` directory of the CFX-5 installation on the master node, and be made readable by all users of the software. This file is a database containing information about where CFX-5, and consequently the PVM software, have been installed on each system. For details please see **PVMhosts File (p. 90 in Installing & Introduction to CFX-5)**.
- Another file named **`hosts.ccl`** must also exist in the `<CFXROOT>/config/<release>` directory on the master node, and be made readable by all users of the software..This file is a database containing information about the available nodes and where CFX-5 has been installed on each of them. Please see **hosts.ccl File (p. 91 in Installing & Introduction to CFX-5)** for further details.
- If you are running machines as slave nodes, you should ensure that the rsh service is installed and working on each of those machines (many Windows machines do not already have an rsh service installed). Full details are available in **Setting up the rsh Service (p. 94 in Installing & Introduction to CFX-5)**.
- You can then run your simulation on other machines by clicking the  icon and selecting a host from the list of all the machines which are available for parallel processing (this should have been set up when CFX-5 was installed). To select the hosts you require, you

should click on the name of each host you want to use (hold down the control key to select more than one host) and then click on **Add**. You must make sure that you include the name of the machine that you are currently logged onto. To increase or decrease the number of partitions for a particular system, highlight it and then use the **+** or **-** buttons to increment.



- For Advanced options, check the **Show Advanced Controls** option. When you click on the **Partitioner** tab some new options become available Please see **Partitioner (p. 14)**. The Solver tab will allow you to make further changes. Please see **Solver (p. 17)**.
- Click on **Start Run** to begin the solution process.

The run will then progress through the partitioning, and then onto the solution of your CFD problem. You will find extra information in the Output File for a parallel run, and this is described in **The CFX-5 Output File (p. 106)**.

Partitioning the model only

You should first define your run as described in **How to use the CFX-5 Solver Manager (p. 48)**, steps 1 to 5. Then continue as described below.

1. Select **Partitioner Only** in the **Type of Run** box.
2. Make should check that you have chosen either **`Local Parallel`** or **Distributed Parallel** as the **Run Type**. When the **Partitioner Only** option is selected, if the run mode is **Serial**, it will automatically switch to **Local Parallel**. The number of partitions created in a

Partitioner Only run is equal to the **Number of Processes** if **Run Mode** is **Local Parallel**. If **Run Mode** is **Distributed Parallel**, the number of partitions created is equal to the total number of partitions on the distributed hosts.

3. You may wish to change the partition settings from their default values. Make sure **Show Advanced Controls** is checked and click on the **Partitioner** tab. Please see **Partitioner (p. 14)** for more details.

4. Click on Start Run to begin the partition process.

The file **`<filename>.par`** will be created in the working directory, where **`<filename>.def`** is the name of the current run directory.

Running the CFX-5 Solver on a model which has already been partitioned

You should first select **Parallel** as the Run Type. You should then define your run as described in **How to use the CFX-5 Solver Manager (p. 48)**, steps 1 to 5. Then continue as described below.

1. Make sure that the **Show Advanced Controls** toggle is checked and click on the **Partitioner** tab.

2. In the Partitioner tab menu, select the partitioned file by clicking on the  icon next to **Initial Partition File**. (this will be called **`<filename>.par`** if your Definition File was called **`<filename>.def`**).

3. For further details about the Partitioner tab menu and the Solver tab menu, please see **Partitioner (p. 14)** and **Solver (p. 17)**.

4. Click Start Run to begin the calculation.

The run will then progress as for an ordinary serial run. You will find extra information in the Output File for a parallel run, and this is described in **The CFX-5 Output File (p. 106)**.

Restarting a Run

This section describes how you can calculate some more timesteps on a Run that has already been sent to the CFX-5 Solver. You can either simply restart from where you left off, or change some details of the simulation before continuing.

You may want to continue from where you left off if, for example:

- you stopped the CFX-5 Solver prematurely and now want to continue running.
- your solution did not converge sufficiently within the maximum number of timesteps that you specified, and you want to make the CFX-5 Solver calculate some more timesteps to achieve the convergence.

Instead of restarting from where you left off, you could simply re-run the problem with a larger Maximum Number of Timesteps, for example. However, restarting from where you left off will enable you to avoid wasting time re-calculating results that you already have.

You may want to change some details of the physical models or parameters before restarting if, for example:

- you change your mind about how well converged you require the solution.
- you have been solving for a laminar flow but as the solution emerges you discover that you need to use a turbulence model. See **Changing the Physics in your CFX-Build Model (p. 475)**.
- you have been having difficulty obtaining a converged solution and you are advised to run your problem with a more robust physical model first, then change to the physical model you require after obtaining an approximate solution to your problem. See **Generating Initial Conditions (p. 488)**.

In general you cannot restart a run when the topology or mesh for the problem has changed, but you can interpolate your results onto a new definition file. See **Restarting a Run After Changing the Topology or Mesh (p. 58)**.

Restarting a Run from where it stopped

This is very easy to do using the CFX-5 Solver Manager. Simply open the **Define Run** form and make sure the directory selected under **Working Directory** is set to the directory which contains the CFX-5 Results File and Output File produced by the first Run of the CFX-5 Solver. Then create a **New Run**, specifying the old Results File as the **Input File**. You can then start the CFX-5 Solver as described in steps

6 to 8 in **Using the CFX-5 Solver Manager for a Serial Run (p. 49)**. You do not need to specify an Initial Values File, since the results contained in your Results File will be used as initial values by the CFX-5 Solver.

You may also select Tools>Restart Run if you are viewing the results using the **Monitor Finished Run** function (see **Monitor Finished run (p. 19)**).

You will notice slight differences in the Output File and Residuals Plots, compared to what you would get if you had just allowed the CFX-5 Solver to continue in the first place. The table which is produced for each timestep in the Output File and Text Output Area will look something like the following.

```
================================================================================
 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|
 +----------------------+------+---------+---------+------------------+
```

The first line of each table contains information relevant for both your whole calculation and for the restarted Run only. The whole calculation is the original Run and the new restarted Run taken together. The values in brackets show information relevant to only the restarted Run. In the above table the information shows that this is the first Outer Loop Iteration or Timestep of the restarted run, but the thirtieth iteration of the entire calculation.

The Residual Plots only show the residuals for the restarted Run. A new Output File, which only contains the information relevant for that restart will be created for each restart.

Restarting a Run which uses Mesh Adaption

Restarting a Run which uses Mesh Adaption will have no effect on the final results. If you have specified the maximum number of adaption steps, then the CFX-5 Solver will determine how many adaption steps were completed in the initial run when determining how many adaption steps are remaining.

Restarting a Run after changing the physical model or solver parameters

Restarting a Run after changing its properties is quite different to restarting without changing properties. To do so, you will need to create a new Definition File with the changed properties, and then use the old Results File as an Initial Values File. In this case, only the CCL and adaption parameters are read from the new Definition File. All other information including the mesh and topology are read from the old Results File.

To create a new Definition File, you should reopen your CFX-Build database and change the appropriate parameters from within CFX-Build. You should not change the number or name of boundary conditions (although you can change their type), nor should you change the geometry, the reference pressure, or the name of any predefined Sub-domains. If you change the Heat Transfer Model from None to any other model, you will be required to enter the specific heat capacity and the thermal conductivity of the fluid. On the **Initial Values** or **Initial Guess** form in CFX-Build, you should set the initial values of the variables contained in your old Results File to **Read From File**.

You can run the CFX-5 Solver on your new Run by setting **Input File** to be the newly-created Definition File, and the **Initial Values File** to the old Results File, on the **Define Run** form.

See **Introduction to Initial Guess and Initial Values (p. 1177 in CFX-Build Chapter 8)** for more information on the **Initial Values** form, and **Using the CFX-5 Solver Manager for a Serial Run (p. 49)** for more details about using an Initial Values File on the **Define** form in the CFX-5 Solver Manager.

Restarting a Run After Changing the Topology or Mesh

If you change the geometry, connectivity, mesh or boundary condition names/location you will not be able to use an old results file to restart your run. The following are examples that would change the topology and/or mesh:

- If your mesh is re-created using different parameters, mesh controls or inflated boundaries.
- If the underlying geometry has changed.
- If the connectivity of the geometry has changed (e.g. specifying domains or sub-domains differently).
- If a new domain or sub-domain is added.
- You cannot change the name or location of any boundary conditions (including the Default boundary condition). However, you can change the boundary condition type.

To restart a run after making these types of changes you should interpolate your results from the old results file onto the new definition file, see **Interpolate (p. 41)**. This will not produce a “clean” restart and you should expect to see a jump in the residual levels upon restarting.

Memory and CPU Requirements

This section provides information on typical increases in CPU time and memory requirements incurred by some simulations and physical models.

Tetrahedral Mesh

The ratio of elements to nodes is approximately 5:1 for a tetrahedral mesh. For example, if there are 5 million tetrahedral elements 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.

The 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 is about 2 times the memory required for a hexahedral mesh of the same number of nodes.

Special Solver Executables

On all systems there are default solver executables. On some systems there are additional solver executables. There are three classes of special executables: “64-bit”, “double precision” and “chip-specific”. Not all of these special executables are offered on all systems (see **Supported Platforms (p. 29 in Installing & Introduction to CFX-5)**).

64-bit

“64-bit” solver executables permit individual solver processes in excess of 2 gigabytes (generally the default solver executables are “32-bit” executables, which are limited to at most 2 gigabytes of RAM, or slightly less depending on the computer vendor). There is generally no RAM penalty or CPU time penalty with “64-bit” executables (the basic floating point numbers are still stored using 32 bit words, only the addressing of data involves 64 bit addresses).

Double Precision

Double Precision solver executables store basic floating point numbers as 64 bit words. These executables are available to permit more accurate numerical mathematical operations. Double precision accuracy might be needed if the computational domain involves a huge variation in grid dimension, aspect ratio, pressure range, etc.

When double precision is used, the computer memory used for a given problem and grid size is double that of the default (single precision) solver executable. Stated another way: the maximum problem size that you can run on a given computer for the double precision solver executable is half that of the default single precision solver executable.

Chip-specific

Chip-specific solver executables have been compiled taking specific advantage of the a computer chips architecture (e.g. for a Pentium III chip set only). In contrast, the default solver executables are compiled in such a way as to run on a family of chip sets (e.g. all Pentium class chips). There is no penalty in terms of memory with the chip-specific solver executables, and generally they provide a performance improvement of at least 25% compared to the default solver executables. No user choice is required. If a chip-specific solver is available for the hardware you are using, it will automatically be detected and used.

Turbulence

Zero Equation Models

The use of this model will only incur a small increase in CPU time and memory requirements compared to laminar flow.

Two Equation Models

Two additional scalar equations are solved for two equation turbulence models. The SST model will have a slight additional cost over other two equation models since a wall scale equation is also solved. Consider the case of single phase, single component laminar flow in which the U-Mom, V-Mom, W-Mom and P-Mass equations are solved. You can expect a CPU cost increase of the order of 50% by the addition of a two-equation turbulence model. Memory requirement increases are small.

Reynolds Stress Models

This model adds six scalar equations for each of the Reynolds Stresses as well as the Eddy Dissipation equation. It will be approximately 2.5 times more expensive compared to 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. You can expect a CPU cost increase of the order of 120% by the addition of a Reynolds Stress turbulence model. Memory requirement increases are small.

Energy

Both the Thermal and Total Energy models require the solution of an additional scalar equation. The solution of the energy equation typically requires 1/3 of the CPU required for the U-Mom, V-Mom, W-Mom and P-Mass equations. Memory requirement increases are small.

CHT

Only the energy equation is solved in CHT regions, so compared to the same number of nodes in a fluid region CPU costs will be much less (U-Mom, V-Mom, W-Mom and P-Mass are not solved).

Multicomponent Flows

Each additional components adds an extra scalar equation, therefore as you increase the number of components the CPU time required to solve the Mass Fraction equation will increase linearly. You can expect each component to add approximately 25% of the CPU required for the U-Mom, V-Mom, W-Mom and P-Mass equations.

Multiphase Flows

Homogeneous

For 2-phase flow using the Homogeneous model you can 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 is not significant.

Interfluid Transfer

For 2-phase flow using the Particle or Mixture models you can 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 is not significant.

N-Phase Flow

As the number of fluids increases you can expect memory and CPU requirements to increase approximately linearly for small N. Tetrahedral meshes will be closer to 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 Increase (Hex Mesh)	Memory Increase (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

You can expect the CPU factor to be slightly less than the corresponding memory factors, but the trend will be the same.

Additional Variables

This adds a single scalar equation for each additional variable. You can expect a CPU cost increase of approximately 20% for each additional variable over the solution of the U-Mom, V-Mom, W-Mom and P-Mass equations for a single-phase single-component case. Increases in memory requirements are small.

Combustion

Modelling combustion will incur slightly higher cost increase compared to multicomponent flow with the same number of components. For multi-step reactions each component is solved using the coupled solver, this will incur additional CPU time which will not increase linearly with the number of components.

Radiation

This adds a single scalar equation. Cost increases will be similar to the Energy equation.

GGI Interfaces

An intersection algorithm which is performed at the start of a simulation to connect each side of a GGI connection will incur a one-time cost.

Each GGI connection adds approximately an additional 5% of CPU time and memory required to a simulation, compared to one without any GGI connections. This number can vary greatly, as it is a function of the number of nodes involved in a GGI connection, compared to the number of nodes that are not involved in the GGI connection. There is also a dependence on the geometric complexity of the GGI connection.

Transient

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

Residual plotting

The CFX-5 Solver calculates the solution to various equations given the appropriate boundary conditions and models for your particular CFD problem. These equations are described fully in **Governing Equations (p. 262)**. At any stage of your calculation, each equation will not be satisfied exactly, and the “residual” of an equation tells you by how much the left-hand-side of the equation differs from the right-hand-side at any point in space. If your solution is “exact” then the residuals will all be zero. (By “exact”, we mean that each of the relevant finite volume equations is satisfied precisely; however, since these equations only model the physics approximately, this does not mean that your solution will exactly match what happens in reality.) If your solution is converging, the residuals should be decreasing for successive timesteps.

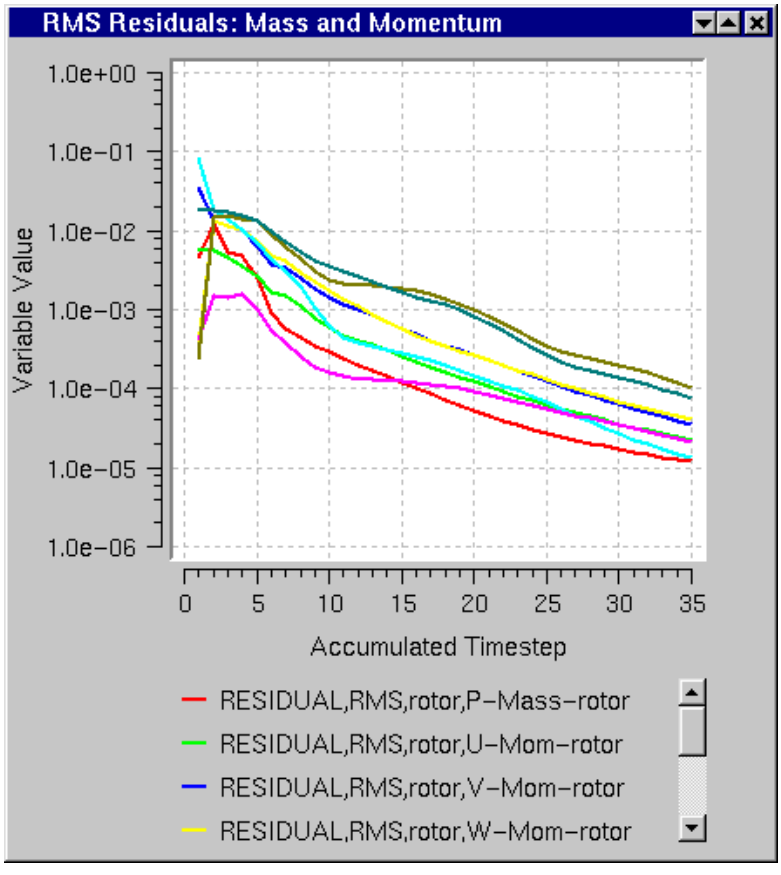
If you are told, for example, that a given residual is 0.0005 kg s^-1, it is not obvious whether such a residual is large or small. For instance, if your problem involves flows such that about 0.5 kg flows into (and out of) each mesh element every second, then a residual of 0.0005 kg s^-1 means that your equation is satisfied to within one part in a thousand, which would be a reasonable solution. However, if your 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 your solution is not good. To make the scales of the residuals meaningful, the solver normalizes their values by dividing by appropriate scales at each point. It is these Normalised Residuals that are plotted by the solver manager using a log (base 10) scale.

The exact details of how the residuals are normalised is complicated, more details are given in **Residual Normalisation Procedure (p. 341)**. However, it is useful to know that the solution field contributes to the normalisation. Hence the residuals for the first time step, or after a failure of the linear solver may be meaningless.

A measure of how well your solution is converged can be obtained by plotting the residuals for each equation at the end of each timestep. For a well-converged solution, you would expect the maximum residual to be around 10^-3; typically the RMS residual will be an order of magnitude lower than this. (The RMS (Root Mean Square) 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; it should give you an idea of a typical magnitude of the residuals.) You

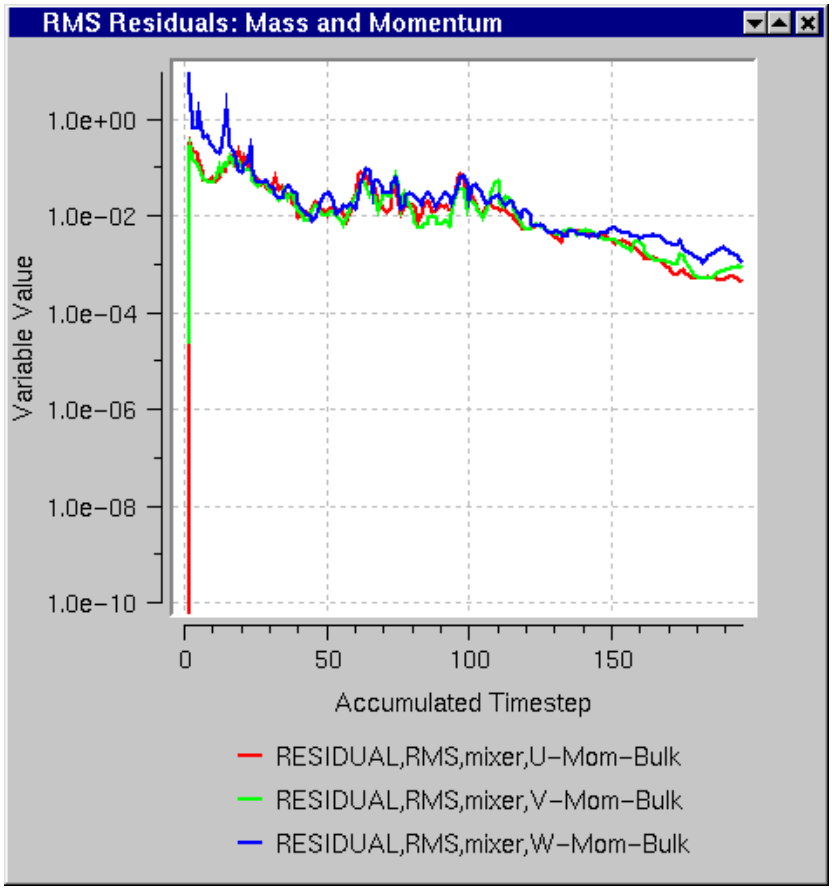
can display the Maximum Residuals and/or the RMS Residuals in the Graphics Area by selecting the appropriate toggle in the Monitor Settings form which can be found in Workspace Properties in the Workspace menu.

The increase of a residual after any particular timestep does not imply that your solution is diverging. It is usual for residuals to occasionally get larger, especially at the beginning of a Run. A typical Residual Plot of a Run which has converged quickly is shown below.



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

Next, we show a Residual Plot for a run that did not converge to the required level very quickly. It is clear that although there is an overall trend of the residuals falling, convergence in this case has taken longer.



It is also possible to have runs that do not converge at all, but simply deviate around the same values. Such an output would resemble the pattern in the above graph between 0 and 100 timesteps, without converging further after many more timesteps.

A Residual Plot where the solution actually diverged is not shown, because it is more usual in this case for the CFX-5 Solver to fail.

If you are having trouble converging, or convergence is only happening very slowly, some tips on how to improve the convergence are given in **Monitoring Convergence (p. 498)**.

If you want to obtain Residual Plots for old Runs, simply select **File>Monitor Finished Run** and select the file you wish to view.

Starting the CFX-5 Solver from the Command Line

The CFX-5 Solver is a separate module of CFX-5 which has no graphical user interface. You can start it in three ways: using the  icon, selecting **File>Define Run** in the CFX-5 Solver Manager (see **Define Run (p. 11)**) or directly from the command line. If you use the CFX-5 Solver Manager to start the CFX-5 Solver, then you will be able to see some details of your solution as it emerges. The graphical user interface of the CFX-5 Solver Manager allows you to set various options and to control the process more easily. However, if you want to run the CFX-5 Solver in batch mode, you can start it from the command line, and this is described in this section.

To start the CFX-5 Solver from the command line, you will need to type

```
cfx5solve -def <file>
```

(where `<file>` is the name of your Definition File) in a UNIX terminal or a suitable Windows command line and press the Return key. (Windows command lines are discussed in **Command line (p. 261 in Installation & Introduction to CFX-5)**.)

You can also start the CFX-5 Solver Manager from the command line To do this, you also need to use the `cfx5solve` command. You will need to type

```
cfx5solve
```

and press Return.

However, there are also various other options which you can use with the `cfx5solve` command. The full command is given below, there are two alternative forms, one which starts the CFX-5 Solver and one which starts the CFX-5 Solver Manager (you are advised to check these for the latest changes by typing `cfx5solve -help`):

```
cfx5solve -definition <file> [-help] [-initial <file>]
[-double] [-single]
[-nosave|-save] [-name <name>]  [-size #]
[-solver <executable>] [-partition <#partitions>]
[-parallel] [-hstfile <file>] [-parfile <file>] [-serial]
[-verbose]
```

or

```
cfx5solve [-interactive [-definition <file>]]
```

```
[-display <display>] [-help] [-noautohelp]
[-solver <executable>] [-verbose]
```

where `[]` denotes an optional argument, `|` separates mutually exclusive arguments and `< >` denotes that substitution of a suitable value is required. All other arguments are keywords, some of which have a short form.

Some of the various arguments are briefly described in the table below; execute `cfx5solve -help` to view a full list of available arguments.

Argument	Alternative form	Usage
-batch		Run the CFX-5 Solver in batch mode (i.e. without starting the CFX-5 Solver Manager. This is the default mode.
-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 definition file for the run, the command language contained in that definition file will take precedence over that supplied. See also -ccl.
-ccl <file>		Reads additional Command Language from the named file. This will override any Command Command Language specified in the Definition File.
-check-only		When running in batch mode, this will cause cfx5solve to verify its arguments, but exit before starting any processes, and is mostly for use by the CFX-5 Solver Manager.
-database <file>	-db <file> -adapt-db <file> -refiner-db <file>	Identify the CFX-Build database file that is required when adapting to the geometry. If the database file is not identified, mesh adaption will occur using the existing mesh, regardless of the settings in the CFX-Build definition file.
-definition <file>	-def <file>	Use `<file>` as the input file, which may be a CFX-5 Definition File or a Results File for a restart. The file specified is used in the same way as the Input File on the **Define Run** form; see **Define Run (p. 11)** for more details.
-display <display>		Use the X11 server `<display>` instead of the X11 server defined by the DISPLAY environment variable. (UNIX only)
-double		Runs the double precision version of the CFX-5 Solver. See -single.
-example <file>	-eg <file>	Runs the CFX-5 Solver on one of the CFX-5 Example definition files provided with the product. The option static_mixer is currently available.

Argument	Alternative form	Usage
-help	-h	Prints a message containing this information.
-hstfile <file>		Use <file> as the host file for a parallel run
-initial <file>	-ini <file>	Use the initial values in the CFX-5 Results File `<file>`. This option may only be used together with a CFX-5 Definition File which was created with the “Read All from File” selected on the Initial Values form in CFX-Build, or with the initial value of any variables set to “Read from File”, or with mesh generation turned OFF. The file specified is used in the same way as the Initial Values File on the **Define Run** form, see **Define Run (p. 11)** for more details.
-interactive	-int -manager	Run the interactive CFX-5 Solver Manager to start a new run and/or manage or monitor an existing runs.
-monitor <file>		When starting the Solver Manager, monitor the run represented by <file>, which may be a CFX-5 Results File or Output File
-name <name>		Choose names for the exported files and temporary directory based on the problem name `<name>` instead of the Definition File name unless other names are explicitly defined. (Note: This name cannot be set when using the CFX-5 Solver Manager to start the CFX-5 Solver.)
-noautohelp		By default the server for the on-line help system is started automatically at the same time as the CFX-5 Solver Manager, to reduce the time taken for the help text to appear when you first request help. If you know you are unlikely to use the on-line help and you want to reduce memory usage you can set this option, in which case the server will be started when you choose an item on the Help menu. (UNIX only)
-parallel	-par	Runs the solver in parallel mode. This option can be combined with the `-partition` option for a partitioning run.
-parallel-mode		Uses the named parallel mode. Currently valid modes are **pvm** and **mpi**.
-par-l-ocal		Allows fast setup of a parallel run on the the local host only.
-par-dist		Allows fast setup of a distributed parallel run. See the examples below.

Argument	Alternative form	Usage
-par-host-list <host1>,[<host2>,...]		When running in parallel, do not use the list of hosts in the host file passed in through the -hstfile argument. Instead, use the list from this argument. If the -hstfile argument is also given, host details will be taken from that file if possible; if the -hstfile argument is not given, or if any of the specified hosts are not available in the file so named, details are read from the hosts.ccl file which is also used by the Solver Manager.
-parfile <file>		Use `<file>` as the partitioning information file for a partitioning or parallel run.
-parfile-read <parfile>		Specifies the name of an input partition file used to set up a partitioning or parallel run
-parfile-write <parfile>		Specifies the name of a partition file to write to, containing the information from a partitioning run
-partition <#partitions>	-part <#partitions>	Runs the solver in partitioning mode. The specified <#partitions> must also be specified to a dummy value if a par-file is also specified and the required number of partitions is taken from this file.
-part-only <# of partitions>		Runs the solver in partitioning mode only. This is normally equivalent to -part, but may be necessary if partitioning a results file from a previous run.
-part-mode <mode>		Sets the partitioning mode to use when running the partitioner. Valid options are metis-kway, metis-res, rcb Finer control over the partitioning method is available through the Command Language.
-priority <number>	-pri <number>	Allows the specification of a job priority to a solver run. Priority 2 is the default priority setting, with priority 1 being lower than the default. High priorities can be set with 3 and 4.
-save		Do not delete any temporary files after the run. Normally the standard temporary files created by CFX-5 Solver are deleted automatically after each run.
-serial		If restarting from a results file produced by a parallel run, this forces the run to be done in serial.
-single		Runs the single-precision version of the CFX-5 Solver. This is the default, but is provided for overriding any information which might be stored in the CFX Command Language dataset in a file from previous runs. See -double.

Argument	Alternative form	Usage
-size <factor>	-s <factor>	Change the memory estimates used by the CFX-5 Solver by a factor of <factor>. By default the memory estimates contained in the CFX-5 Definition File are used. Sometimes these are inaccurate and this option needs to be used to increase the memory allocated. For example using `-size 1.05` will increase the memory allocated by 5%. This option provides the same functionality as set on the **Solver Memory** form, which is described in **Solver (p. 17)**. Further options for controlling the CFX-5 Solver memory allocation are available, execute **`cfx5solve -help`** for full details.
-size-part <factor>	-sizepar <factor>	Change the memory estimates used by the partitioner by a factor of <factor>. See also -size. Further options for controlling the partitioner memory allocation are available, execute **`cfx5solve -help`** for full details.
-size-nr <size> -size-ni <size> -size-nd <size> -size-nc <size> -size-nl <size>	-nr <size> -ni <size> -nd <size> -nc <size> -nl <size>	These flags are for advanced users to change the memory allocation parameters for the solver. Usually, you should use the -size argument instead. <size> is the desired memory allocation in words, and may have K or M appended for kilo- or mega-.
-size-part-nr <size> -size-part-ni <size> -size-part-nd <size> -size-part-nc <size> -size-part-nl <size>	-nrpar <size> -nipar <size> -ndpar <size> -ncpar <size> -nlpar <size>	This is the same as the -size-* arguments above, but give the sizes needed for the partitioner rather than the solver.
-solver-arch <arch>		Specifies an architecture string, which is used to locate the version of the CFX-5 Solver to run if optimised solver . The <arch> should be a valid CFX-5 architecture string, and the software expects to find a version of solver.exe in <CFX_ROOT>/bin/<version>/<arch>/solver.exe
-verbose	-v	Specifying this option may result in additional output being sent to the standard Output File (normally the screen).

More details on the command line options `-help`, `-display` and `-verbose`, together with a more detailed description on how to use command line options, are given in **CFX-5 Commands (p. 161 in Installing & Introduction to CFX-5)**.

Some examples are given below.

1. To start the CFX-5 Solver running from the Definition File model.def, type:

```
cfx5solve -def model.def
```

2. To start the CFX-5 Solver Manager running, passing it the name of the Definition File, type:

```
cfx5solve -interactive -def model.def
```

3. To produce a partition file `model.par` with the MeTiS partitioning method and 7 partitions, but NOT to run the CFX-5 Solver to solve for the solution, make sure that no file `model.par` exists in the working directory and type:

```
cfx5solve -def model.def -partition 7
```

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 haven't specified the file `model.par`. Because this could potentially be confusing, you are advised to use the CFX-5 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.

4. To run the CFX-5 Solver in distributed parallel, starting from the Definition File `model.def`, and using 1 processor on `host1`, 1 processor on `host2` and 2 processors on `host3`, type:

```
cfx5solve -def model.def -par-dist
'host1,host2,host3*2' -partition
```

   The hosts `host1`, `host2` and `host3` must be defined in the central `hosts.ccl` file.

5. To run the CFX-5 Solver in parallel, starting from the Definition File model.def and running only on the local machine with 2 partitions, type:

```
cfx5solve -def model.def -par-local -partition 2
```

   Note that this method does not require you to create a `hst` file.

6. 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-5 Solver by typing:

```
cfx5solve -def model.def -parallel -parfile
model.par
```

cfx5stop

After the CFX-5 Solver is running, you can stop it by typing **cfx5stop** at the command line. Suppose that your Run is called **mixer_001** in the current directory. Then there will be a temporary directory called `mixer_001.dir` in the current directory while that Run is actually running. To stop the Run, you need to enter the following command line:

```
cfx5stop -directory mixer_001.dir
```