ANSYS CFX Tutorials

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

Release 13.0
November 2010

Copyright and Trademark Information

Disclaimer Notice

U.S. Government Rights

Third-Party Software

Published in the U.S.A.

Table of Contents

Chapter 1: Introduction to the CFX Tutorials

The CFX tutorials are designed to introduce general techniques used in CFX and to provide tips on advanced modeling.

The initial tutorials introduce general principles used in CFX, including setting up the physical models, running CFX-Solver and visualizing the results in CFD-Post; the later tutorials highlight specialized features of CFX.

You should review the following topics before attempting to start a tutorial for the first time:

1.1. Preparing a Working Directory and Starting ANSYS CFX in Standalone Mode

ANSYS CFX uses a working directory as the default location for loading and saving files for a particular session or project. Before you run a tutorial, you must create a working directory.

- If you plan to run through the whole tutorial, copy the files that are listed near the beginning of the tutorial to your working directory. This practice will prevent you from making accidental changes to any of the files that came with your installation. The files are available from your CFX installation and from the ANSYS Customer Portal.

  The tutorial input files are available in your CFX installation in `<CFXROOT>/examples` and `<CFX-ROOT>/etc/model-templates`, where <CFXROOT> is the installation directory for ANSYS CFX.

  The tutorial input files are available from the ANSYS Customer Portal (www.ansys.com/customerportal/) by using the Download Wizard to download the `ANSYS_Fluid_Dynamics_Tutorial_Inputs.zip` file. Once downloaded and expanded, the tutorial input files can be found in the following locations: `<PATH>\v130\Tutorial_Inputs\Fluid_Dynamics\CFX\examples` and `<PATH>\v130\Tutorial_Inputs\Fluid_Dynamics\CFX\etc\model-templates`, where <PATH> is the directory where the downloaded `.zip` file was expanded.
- If you plan to run the provided tutorial session file (so that you can immediately run the simulation in CFX-Solver), there is no need to copy any files to your working directory; all required files that are missing from the working directory are copied automatically when you play the session file. Note that any pre-existing files in your working directory that are inconsistent with either the tutorial or the current version of the software will not be overwritten and could produce unexpected results.

Before you start CFX-Pre, CFX-Solver Manager, or CFD-Post, set the working directory. The procedure for setting the working directory and starting ANSYS CFX in Standalone is listed below:

1. Start the ANSYS CFX Launcher.

You can start the ANSYS CFX Launcher in any of the following ways:

- On Windows:
  - From the **Start** menu, select **All Programs > ANSYS 13.0 > Fluid Dynamics > CFX**.
  - In a DOS window that has its path set up correctly to run CFX, enter `cfx5` (otherwise, you will need to type the full pathname of the `cfx5` command).
- On UNIX, enter `cfx5` in a terminal window that has its path set up to run CFX.

2. Specify the **Working Directory** on the ANSYS CFX Launcher window.
3. Click the **CFX-Pre 13.0** button.
4. If you were directed here at some point during a tutorial, return to that location.

Note

All tutorials assume that the **CFX run history and multi-configuration options**, under the **Load Results File** dialog box in CFD-Post, is set to **Load only the last results**.

1.2. Running ANSYS CFX Tutorials in ANSYS Workbench

Most ANSYS CFX tutorials are written to work in Standalone Mode. This section includes the steps required to run these tutorials in ANSYS Workbench:

Tip

You may find it useful to open the ANSYS CFX help from the ANSYS CFX Launcher (which does not take up a license).

1.2.1. Starting CFX-Pre

1. Start ANSYS Workbench.
   - To launch ANSYS Workbench on Windows, click the **Start** menu, then select **All Programs > ANSYS 13.0 > Workbench**.
   - To launch ANSYS Workbench on Linux, open a command line interface, type the path to "runwb2" (for example, "~/ansys_inc/v130/Framework/bin/Linux64/runwb2"), then press **Enter**.
2. From the tool bar, click **Save As** and use the **Save in** field to set the directory to which you want to save the project file. This directory will be referred to as the working directory. Set the project name in the **File name** field and click **Save**.

3. In the **Toolbox** pane, open **Component Systems** and double-click **CFX**. A **CFX** system opens in the **Project Schematic**.

   Note

   You use a **CFX** component system because you are starting with a mesh. If you want to create the geometry and mesh, you will start with a **Fluid Flow (CFX)** system.

4. Type in the new name, such as *System 1*, to replace the highlighted text below the system.

   Alternatively, you can right-click the first cell in the system and select **Rename**. The name will be highlighted. Now you can change the highlighted text by typing in the new name.

5. In the **Project Schematic**, right-click the **Setup** cell and select **Edit** to launch CFX-Pre.
6. Continue from the *Defining a Simulation in CFX-Pre* section of your tutorial.

1.2.2. Writing the CFX-Solver Input (.def) File

When running ANSYS CFX within ANSYS Workbench, no action is required for this section of the tutorial. The required files are automatically transferred between the cells within the **CFX** component system. Continue from *Obtaining a Solution Using CFX-Solver Manager* (p. 3).

1.2.3. Obtaining a Solution Using CFX-Solver Manager

Once the simulation setup is complete, the **Solution** cell prompts you to refresh it. To refresh that cell:

- Right-click the **Solution** cell and select **Refresh**.

  Note

  If the **Solution** cell displays a prompt to perform an update, ignore it and proceed to the next step.

To obtain a solution, you need to launch the CFX-Solver Manager and subsequently use it to start the solver:

1. Double-click the ANSYS Workbench **Solution** cell. The CFX-Solver Manager appears with the **Define Run** dialog box displayed.
2. Continue from the *Obtaining a Solution Using CFX-Solver Manager* section of the tutorial.

1.2.4. Viewing the Results in CFD-Post

When CFX-Solver has finished, a completion message appears in a dialog box. Click **OK**.

Alternatively, a message saying *This run of the ANSYS CFX-Solver has finished* is displayed in the final line of the `*.out` file in the CFX-Solver Manager.

Once CFX-Solver has finished, you can use CFD-Post to review the finished results. At this point, the **Results** cell in ANSYS Workbench prompts you to refresh:

1. Right-click the **Results** cell and select **Refresh**.
2. When the refresh is complete, double-click the **Results** cell to open CFD-Post.

3. Continue from the *Viewing the Results in CFD-Post* section of the tutorial.

If this is the final section of your tutorial, continue from *Closing the Applications* (p. 4). If you are Running ANSYS CFX Tutorials with a Sequence of Multiple Simulations continue from *Creating CFX Component Systems for Multiple Simulations (As Required)* (p. 4).

1.2.5. Creating CFX Component Systems for Multiple Simulations (As Required)

Now that you have set the physics in the initial state, you will duplicate the *CFX* component system created earlier and edit the physics in the new system. To duplicate the existing **CFX** component system:

1. In the **ANSYS Workbench Project Schematic**, right-click the first cell in the *System 1* system and select **Duplicate**.

   A new system named *Copy of System 1* will appear in the **Project Schematic**.
2. Type in the new name *System 2* to replace the highlighted text below the system.
3. Click the **Solution** cell of the *System 1* and drag it to the **Solution** cell of the *System 2*.

   You will now see a line, indicating a transfer connection, going from **Solution** cell of the *System 1* to the **Solution** cell of the *System 2*.
4. Once, you have set up the new **CFX** component system, continue from Step 5 of *Starting CFX-Pre* (p. 2).

Note

In the tutorial, ignore the steps that tell you to set the initial values file in the **Define Run** dialog box for CFX-Solver Manager. Dragging the solution cell between systems automatically sets the initialization options in CFX-Solver Manager.

1.2.6. Closing the Applications

Close ANSYS Workbench (and the applications it launched) by selecting **File** > **Exit** from ANSYS Workbench. ANSYS Workbench prompts you to save all your project files.

1.3. Playing a Tutorial Session File

Every tutorial involves instructions for setting up a simulation in CFX-Pre. If you want to skip past those instructions and have CFX-Pre set up the simulation automatically, you can run the tutorial session file specified in the tutorial.[1]

Note

Session files and tutorial session files can be played only in ANSYS CFX standalone, not in ANSYS Workbench.

To play a tutorial session file:

1. If required, launch CFX-Pre.

[1] Some tutorials have more than one tutorial session file; each covers a particular set of CFX-Pre setup instructions.

2. Select **Session** > **Play Tutorial**.
3. Select the required tutorial session file (*filename*.pre) for the tutorial. This file is located in <CFX-ROOT>/examples and <CFXROOT>/etc/model-templates, where <CFXROOT> is the installation directory for ANSYS CFX.
4. Click **Open**.
5. If an **Information** dialog box appears, click **OK**. CFX-Pre writes a CFX-Solver input file (*filename*.def) in the directory set as your **Working Directory**. This file is written in the background while CFX-Pre remains open.
6. On the CFX-Pre menu bar, select **File** > **Quit**.
7. On the **ANSYS CFX-13.0 Launcher** click the **CFX-Solver Manager 13.0** button.
8. On the **CFX-Solver Manager** menu bar, select **File** > **Define Run**. The **Define Run** dialog box appears.
9. On the **Define Run** dialog box, click the browse button  . In the **CFX-Solver File** dialog box that appears, choose the *filename*.def file and click **Open**.
10. If you were directed here at some point during a tutorial, return to that location.

Note

Playing a session file may change the default settings under the **Case Options** > **General Options** — these changes will be retained until the case is closed. To override these changes (for example, to **Automatic Default Domain**, **Automatic Default Interfaces**, etc.) the settings must be changed from the **Outline** tree view under **Case Options** > **General Options** rather than from the global options (**Edit** > **Options**). Changes made to the global options are persistent and will not take effect until a new case is opened.

1.4. Changing the Display Colors

If viewing objects in ANSYS CFX becomes difficult due to contrast with the background, you can change the colors for improved viewing. The color options are set in different places, depending on how you run CFX:

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

   The **Options** dialog box appears.
2. Adjust the color settings under CFX-Pre > Graphics Style (for CFX-Pre) or CFD-Post > Viewer (for CFD-Post).
3. Click **OK**.
4. If you were directed here at some point during a tutorial, return to that location.

1.5. Editor Buttons

The ANSYS CFX interface uses editors to enter the data required to set up and post-process a simulation. The editors have standard buttons, which are described next:

- **Apply** applies the information contained within all the tabs of an editor.
- **OK** is the same as **Apply**, except that the editor automatically closes.
- **Cancel** and **Close** both close the editor without applying or saving any changes.

- **Reset** returns the settings for the object to those stored in the database for all the tabs. The settings are stored in the database each time the **Apply** button is clicked.
- **Defaults** restores the system default settings for all the tabs of the edited object.

1.6. Using Help

To invoke the help browser, select **Help** > **Contents**.

You may also try using context-sensitive help. Context-sensitive help is provided for many of the details views and other parts of the interface. To invoke the context-sensitive help for a particular details view or other feature, ensure that the feature is active, place the mouse pointer over it, then press **F1**. Not every area of the interface supports context-sensitive help. If context-sensitive help is not available for the feature of interest, select **Help** > **Contents** and try using the search or index features in the help browser.

Chapter 2: Simulating Flow in a Static Mixer Using CFX in Standalone Mode

This tutorial includes:

This tutorial simulates a static mixer consisting of two inlet pipes delivering water into a mixing vessel; the water exits through an outlet pipe. A general workflow is established for analyzing the flow of fluid into and out of a mixer.

2.1. Tutorial Features

In this tutorial you will learn about:

- Using Quick Setup mode in CFX-Pre to set up a problem.
- Using CFX-Solver Manager to obtain a solution.
- Modifying the outline plot in CFD-Post.
- Using streamlines in CFD-Post to trace the flow field from a point.
- Viewing temperature using colored planes and contours in CFD-Post.
- Creating an animation and saving it as a movie file.

Component	Feature	Details
CFX-Pre	User Mode	Quick Setup Wizard
	Analysis Type	Steady State
	Fluid Type	General Fluid
	Domain Type	Single Domain
	Turbulence Model	k-Epsilon
	Heat Transfer	Thermal Energy
	Boundary Conditions	Inlet (Subsonic)
		Outlet (Subsonic)
		Wall: No-Slip
		Wall: Adiabatic
	Timestep	Physical Time Scale
CFD-Post	Animation	Keyframe

Component	Feature	Details
	Plots	Contour
		Outline Plot (Wireframe)
		Point
		Slice Plane
		Streamline

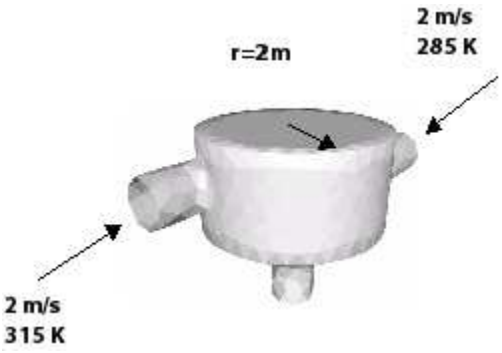
2.2. Overview of the Problem to Solve

This tutorial simulates a static mixer consisting of two inlet pipes delivering water into a mixing vessel; the water exits through an outlet pipe. A general workflow is established for analyzing the flow of fluid into and out of a mixer.

Water enters through both pipes at the same rate but at different temperatures. The first entry is at a rate of 2 m/s and a temperature of 315 K and the second entry is at a rate of 2 m/s at a temperature of 285 K. The radius of the mixer is 2 m.

Your goal in this tutorial is to understand how to use CFX to determine the speed and temperature of the water when it exits the static mixer.

Figure 2.1 Static Mixer with 2 Inlet Pipes and 1 Outlet Pipe



2.3. Before You Begin

Before you begin this tutorial, review the following topics:

- *Preparing a Working Directory and Starting ANSYS CFX in Standalone Mode* (p. 1)
- *Running ANSYS CFX Tutorials in ANSYS Workbench* (p. 2)
- *Changing the Display Colors* (p. 5)
- *Playing a Tutorial Session File* (p. 4)

2.4. Starting CFX-Pre

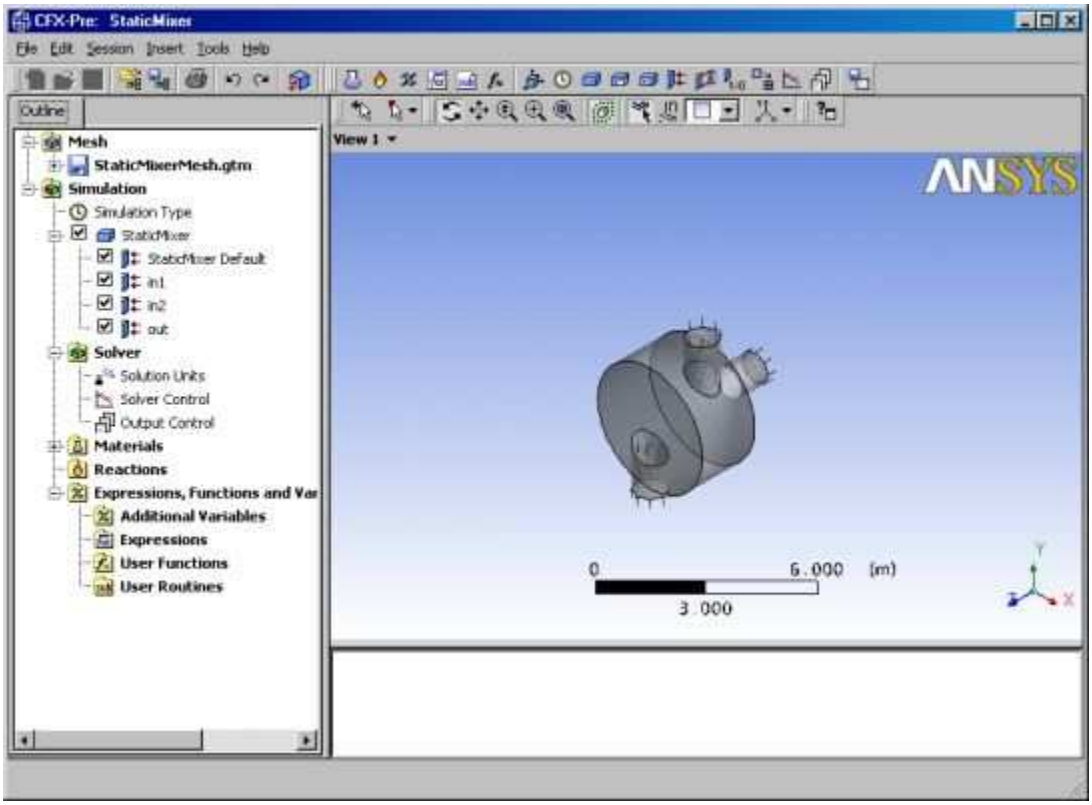
1. Prepare the working directory using the following files in the `examples` directory:
   - `StaticMixerMesh.gtm`
   - `StaticMixer.pre`

2. Set the working directory and start CFX-Pre.

   For details, see *Preparing a Working Directory and Starting ANSYS CFX in Standalone Mode* (p. 1)

2.5. Defining a Case in CFX-Pre

Because you are starting with an existing mesh, you can immediately use CFX-Pre to define the simulation. This is how CFX-Pre will look with the imported mesh:



In the image above, the left pane of CFX-Pre displays the **Outline** workspace. When you double-click items in the **Outline**, the **Outline** editor opens and can be used to create, modify, and view objects.

Note

In this documentation, the details view can also be referenced by the name of the object being edited, followed by the word "details view" (for example, if you double-click the `Wireframe` object, the **Wireframe** details view appears).

2.5.1. Synopsis of Quick Setup Mode

Quick Setup mode provides a simple wizard-like interface for setting up simple cases. This is useful for getting familiar with the basic elements of a CFD problem setup. This section describes using Quick Setup mode to develop a simulation in CFX-Pre.

2.5.2. Workflow Overview

This tutorial follows the general workflow for Quick Setup mode:

2.5.3. Creating a New Case

Before importing and working with a mesh, a simulation needs to be started using Quick Setup mode.

1. In CFX-Pre, select **File** > **New Case**.

   The **New Case File** dialog box is displayed.
2. Select **Quick Setup** and click **OK**.

   Note

   If this is the first time you are running this software, a message box will appear notifying you that automatic generation of the default domain is active. To avoid seeing this message again uncheck **Show This Message Again**.

3. Select **File** > **Save Case As**.
4. Under **File name**, type: `StaticMixer`
5. Click **Save**.

2.5.4. Setting the Physics Definition

You need to specify the fluids used in a simulation. A variety of fluids are already defined as library materials. For this tutorial you will use a prepared fluid, Water, which is defined to be water at 25°C.

1. Ensure that **Simulation Definition** is displayed at the top of the details view.
2. Under **Fluid** select `Water`.

2.5.5. Importing a Mesh

At least one mesh must be imported before physics are applied.

1. In **Simulation Definition**, under **Mesh File**, click *Browse*  .

   The **Import Mesh** dialog box appears.
2. Under **Files of type**, select `CFX Mesh (*gtm *cfx)`.
3. From your working directory, select `StaticMixerMesh.gtm`.
4. Click **Open**.

   The mesh loads.
5. Click **Next**.

2.5.6. Using the Viewer

Now that the mesh is loaded, take a moment to explore how you can use the viewer toolbar to zoom in or out and to rotate the object in the viewer.

2.5.6.1. Using the Zoom Tools

There are several icons available for controlling the level of zoom in the viewer.

1. Click *Zoom Box* 
2. Click and drag a rectangular box over the geometry.
3. Release the mouse button to zoom in on the selection.

   The geometry zoom changes to display the selection at a greater resolution.
4. Click *Fit View*  to re-center and re-scale the geometry.

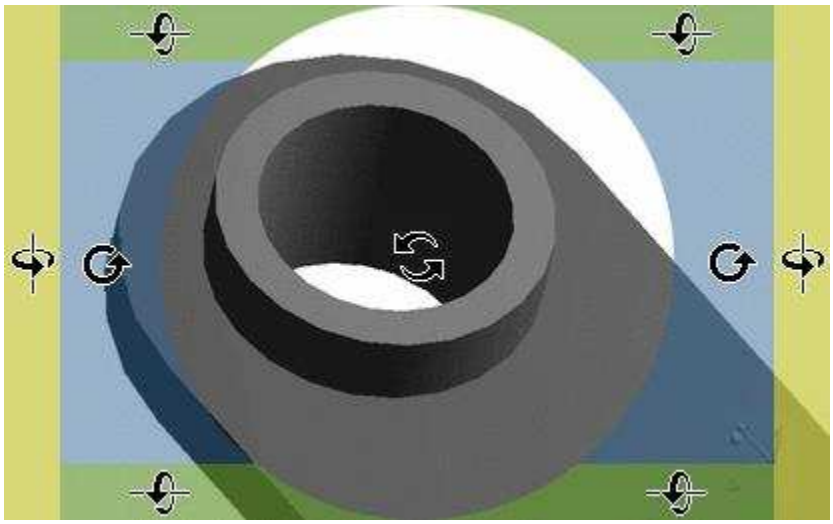
2.5.6.2. Rotating the Geometry

If you need to rotate an object or to view it from a new angle, you can use the viewer toolbar.

1. Click *Rotate*  on the viewer toolbar.
2. Click and drag within the geometry repeatedly to test the rotation of the geometry.

   The geometry rotates based on the direction of movement.

   Notice how the mouse cursor changes depending on where you are in the viewer:



3. Right-click a blank area in the viewer and select **Predefined Camera** > **View From -X**.
4. Right-click a blank area in the viewer and select **Predefined Camera** > **Isometric View (Z Up)**.

   A clearer view of the mesh is displayed.

2.5.7. Defining Model Data

You need to define the type of flow and the physical models to use in the fluid domain.

You will specify the flow as steady state with turbulence and heat transfer. Turbulence is modeled using the k-ε turbulence model and heat transfer using the thermal energy model. The k-ε turbulence model is a commonly used model and is suitable for a wide range of applications. The thermal energy model neglects high speed energy effects and is therefore suitable for low speed flow applications.

1. Ensure that **Physics Definition** is displayed.
2. Under **Model Data**, set **Reference Pressure** to `1 [atm]`.

   All other pressure settings are relative to this reference pressure.
3. Set **Heat Transfer** to `Thermal Energy`.
4. Set **Turbulence** to `k-Epsilon`.
5. Click **Next**.

2.5.8. Defining Boundaries

The CFD model requires the definition of conditions on the boundaries of the domain.

1. Ensure that **Boundary Definition** is displayed.
2. Delete `Inlet` and `Outlet` from the list by right-clicking each and selecting **Delete Boundary**.
3. Right-click in the blank area where `Inlet` and `Outlet` were listed, then select **Add Boundary**.
4. Set **Name** to `in1`.
5. Click **OK**.

   The boundary is created and, when selected, properties related to the boundary are displayed.

2.5.9. Setting Boundary Data

Once boundaries are created, you need to create associated data. Based on *Figure 2.1* (p. 8), you will define the velocity and temperature for the first inlet.

1. Ensure that **Boundary Data** is displayed.
2. Set **Boundary Type** to `Inlet`.
3. Set **Location** to `in1`.

2.5.10. Setting Flow Specification

Once boundary data is defined, the boundary needs to have the flow specification assigned.

1. Ensure that **Flow Specification** is displayed.
2. Set **Option** to `Normal Speed`.
3. Set **Normal Speed** to `2 [m s^-1]`.

2.5.11. Setting Temperature Specification

Once flow specification is defined, the boundary needs to have temperature assigned.

1. Ensure that **Temperature Specification** is displayed.
2. Set **Static Temperature** to `315 [K]`.

2.5.12. Reviewing the Boundary Condition Definitions

Defining the boundary condition for `in1` required several steps. Here the settings are reviewed for accuracy.

Based on *Figure 2.1* (p. 8), the first inlet boundary condition consists of a velocity of 2 m/s and a temperature of 315 K at one of the side inlets.

- Review the boundary `in1` settings for accuracy. They should be as follows:

Tab	Setting	Value
Boundary Data	Boundary Type	Inlet
	Location	in1
Flow Specification	Option	Normal Speed
	Normal Speed	2 [m s^-1]
Temperature Specification	Static Temperature	315 [K]

2.5.13. Creating the Second Inlet Boundary Definition

Based on *Figure 2.1* (p. 8), you know the second inlet boundary condition consists of a velocity of 2 m/s and a temperature of 285 K at one of the side inlets. You will define that now.

1. Under **Boundary Definition**, right-click in the selector area and select **Add Boundary**.
2. Create a new boundary named `in2` with these settings:

Tab	Setting	Value
Boundary Data	Boundary Type	Inlet
	Location	in2
Flow Specification	Option	Normal Speed
	Normal Speed	2 [m s^-1]
Temperature Specification	Static Temperature	285 [K]

2.5.14. Creating the Outlet Boundary Definition

Now that the second inlet boundary has been created, the same concepts can be applied to building the outlet boundary.

1. Create a new boundary named `out` with these settings:

Tab	Setting	Value
Boundary Data	Boundary Type	Outlet
	Location	out
Flow Specification	Option	Average Static Pressure
	Relative Pressure	0 [Pa]

2. Click **Next**.

2.5.15. Moving to General Mode

There are no further boundary conditions that need to be set. All 2D exterior regions that have not been assigned to a boundary condition are automatically assigned to the default boundary condition.

- Set **Operation** to `Enter General Mode` and click **Finish**.

  The three boundary conditions are displayed in the viewer as sets of arrows at the boundary surfaces. Inlet boundary arrows are directed into the domain. Outlet boundary arrows are directed out of the domain.

2.5.16. Setting Solver Control

Solver Control parameters control aspects of the numerical solution generation process.

While an upwind advection scheme is less accurate than other advection schemes, it is also more robust. This advection scheme is suitable for obtaining an initial set of results, but in general should not be used to obtain final accurate results.

The time scale can be calculated automatically by the solver or set manually. The `Automatic` option tends to be conservative, leading to reliable, but often slow, convergence. It is often possible to accelerate convergence by applying a time scale factor or by choosing a manual value that is more aggressive than the `Automatic` option. In this tutorial, you will select a physical time scale, leading to convergence that is twice as fast as the `Automatic` option.

1. Click *Solver Control*  .

2. On the **Basic Settings** tab, set **Advection Scheme** > **Option** to `Upwind`.
3. Set **Convergence Control** > **Fluid Timescale Control** > **Timescale Control** to `Physical Timescale` and set the physical timescale value to `2 [s]`.
4. Click **OK**.

2.5.17. Writing the CFX-Solver Input (.def) File

The simulation file, `StaticMixer.cfx`, contains the simulation definition in a format that can be loaded by CFX-Pre, allowing you to complete (if applicable), restore, and modify the simulation definition. The simulation file differs from the CFX-Solver input file in that it can be saved at any time while defining the simulation.

1. Click *Define Run*  .
2. Set **File name** to `StaticMixer.def`.
3. Click **Save**.

   The CFX-Solver input file (`StaticMixer.def`) is created. CFX-Solver Manager automatically starts and, on the **Define Run** dialog box, the **Solver Input File** is set.
4. If you are notified the file already exists, click **Overwrite**.

   This file is provided in the tutorial directory and may exist in your working directory if you have copied it there.
5. When you are finished, select **File** > **Quit** in CFX-Pre.
6. If prompted, click **Yes** or **Save & Quit** to save `StaticMixer.cfx`.
7. Proceed to *Obtaining a Solution Using CFX-Solver Manager* (p. 16).

2.5.18. Playing the Session File and Starting CFX-Solver Manager

Note

This task is required only if you are starting here with the session file that was provided in the examples directory. If you have performed all the tasks in the previous steps, proceed directly to *Obtaining a Solution Using CFX-Solver Manager* (p. 16).

Events in CFX-Pre can be recorded to a session file and then played back at a later date to drive CFX-Pre. Session files have been created for each tutorial so that the problems can be set up rapidly in CFX-Pre, if desired.

1. If required, launch CFX-Pre.
2. Select **Session** > **Play Tutorial**.
3. Select `StaticMixer.pre`.
4. Click **Open**.

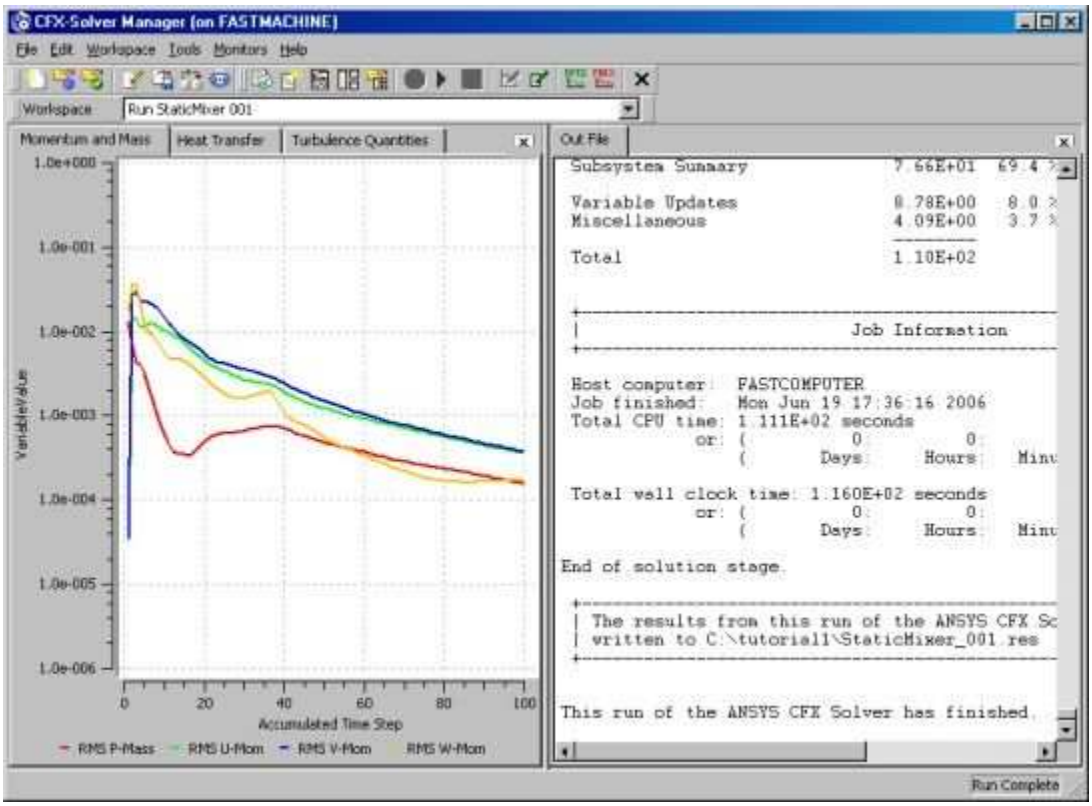
   A CFX-Solver input file is written.
5. Select **File** > **Quit**.
6. Launch the CFX-Solver Manager from CFX Launcher.

7. After the CFX-Solver starts, select **File** > **Define Run**.
8. Under **CFX-Solver Input File**, click *Browse*  .
9. Select `StaticMixer.def`, located in the working directory.
10. Proceed to *Obtaining a Solution Using CFX-Solver Manager* (p. 16).

2.6. Obtaining a Solution Using CFX-Solver Manager

CFX-Solver Manager has a visual interface that displays a variety of results and should be used when plotted data needs to be viewed during problem solving.

Two windows are displayed when CFX-Solver Manager runs. There is an adjustable split between the windows, which is oriented either horizontally or vertically depending on the aspect ratio of the entire CFX-Solver Manager window (also adjustable).



One window shows the convergence history plots and the other displays text output from CFX-Solver.

The text lists physical properties, boundary conditions and various other parameters used or calculated in creating the model. All the text is written to the output file automatically (in this case, `StaticMixer_001.out`).

2.6.1. Start the Run

The **Define Run** dialog box allows configuration of a run for processing by CFX-Solver.

When CFX-Solver Manager is launched automatically from CFX-Pre, all of the information required to perform a new serial run (on a single processor) is entered automatically. You do not need to alter the information

in the **Define Run** dialog box. This is a very quick way to launch into CFX-Solver without having to define settings and values.

1. Ensure that the **Define Run** dialog box is displayed.
2. Click **Start Run**.

   CFX-Solver launches and a split screen appears and displays the results of the run graphically and as text. The panes continue to build as CFX-Solver Manager operates.

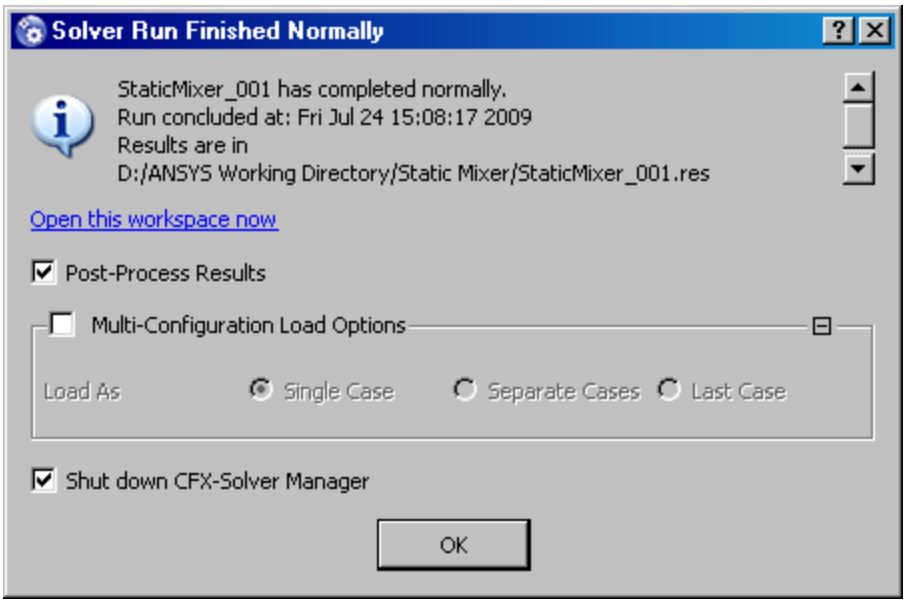
Note

Once the second iteration appears, data begins to plot. Plotting may take a long time depending on the amount of data to process. Let the process run.

2.6.2. Move from CFX-Solver to CFD-Post

Once CFX-Solver has finished, you can use CFD-Post to review the finished results.

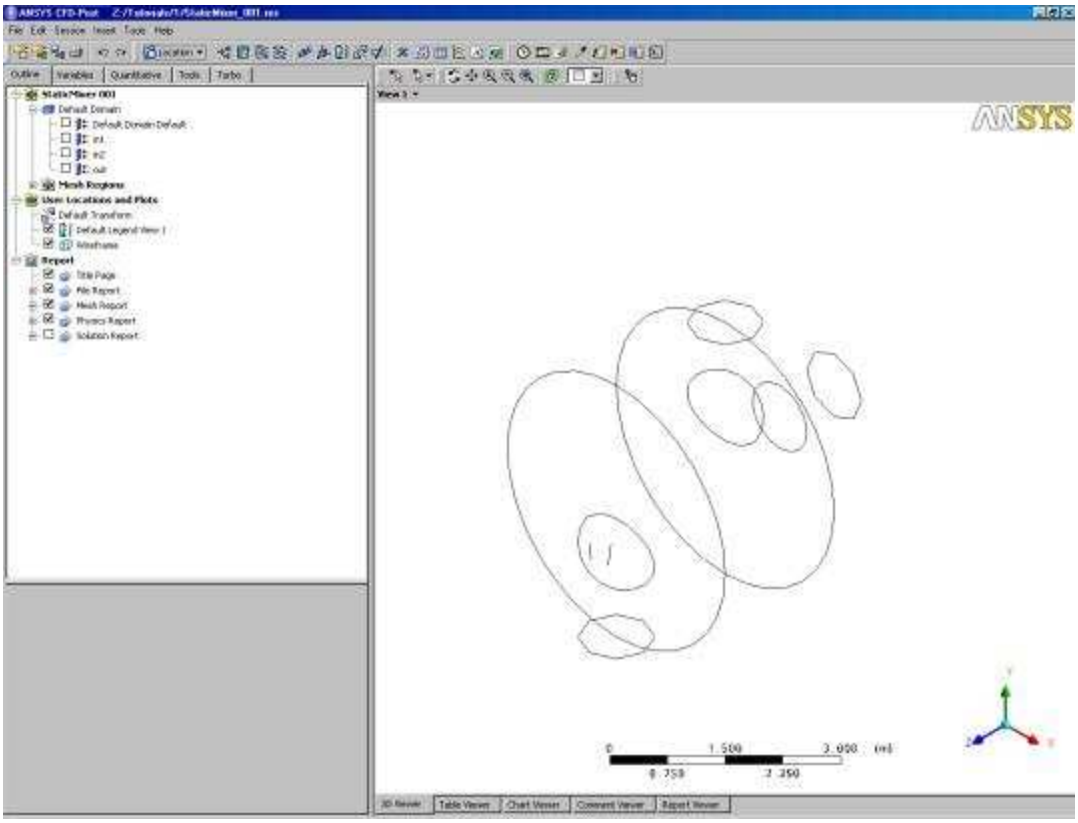
1. When CFX-Solver is finished, select the check box next to **Post-Process Results**.



2. If using Standalone Mode, select the check box next to **Shut down CFX-Solver Manager**.
3. Click **OK**. After a short pause, CFX-Solver Manager closes and CFD-Post opens.

2.7. Viewing the Results in CFD-Post

When CFD-Post starts, the viewer and **Outline** workspace are displayed.



The viewer displays an outline of the geometry and other graphic objects. You can use the mouse or the toolbar icons to manipulate the view, exactly as in CFX-Pre.

2.7.1. Workflow Overview

This tutorial describes the following workflow for viewing results in CFD-Post:

1. *Setting the Edge Angle for a Wireframe Object* (p. 19)
2. *Creating a Point for the Origin of the Streamline* (p. 20)
3. *Creating a Streamline Originating from a Point* (p. 21)
4. *Rearranging the Point* (p. 22)
5. *Configuring a Default Legend* (p. 23)
6. *Creating a Slice Plane* (p. 24)
7. *Defining Slice Plane Geometry* (p. 24)
8. *Configuring Slice Plane Views* (p. 24)
9. *Rendering Slice Planes* (p. 25)
10. *Coloring the Slice Plane* (p. 26)
11. *Moving the Slice Plane* (p. 26)
12. *Adding Contours* (p. 27)
13. *Working with Animations* (p. 28)
14. *Showing the Animation Dialog Box* (p. 28)
15. *Creating the First Keyframe* (p. 29)

16. *Creating the Second Keyframe* (p. 30)
17. *Viewing the Animation* (p. 32)
18. *Modifying the Animation* (p. 32)
19. *Saving a Movie* (p. 33)

2.7.2. Setting the Edge Angle for a Wireframe Object

The outline of the geometry is called the *wireframe* or *outline plot*.

By default, CFD-Post displays only some of the surface mesh. This sometimes means that when you first load your results file, the geometry outline is not displayed clearly. You can control the amount of the surface mesh shown by editing the `Wireframe` object listed in the **Outline** tree view.

The check boxes next to each object name in the **Outline** control the visibility of each object. Currently only the `Wireframe` and `Default Legend` objects have visibility turned on.

The edge angle determines how much of the surface mesh is visible. If the angle between two adjacent faces is greater than the edge angle, then that edge is drawn. If the edge angle is set to 0°, the entire surface mesh is drawn. If the edge angle is large, then only the most significant corner edges of the geometry are drawn.

For this geometry, a setting of approximately 15° lets you view the model location without displaying an excessive amount of the surface mesh.

In this module you can also modify the zoom settings and view of the wireframe.

1. In the **Outline**, under `User Locations and Plots`, double-click `Wireframe`.

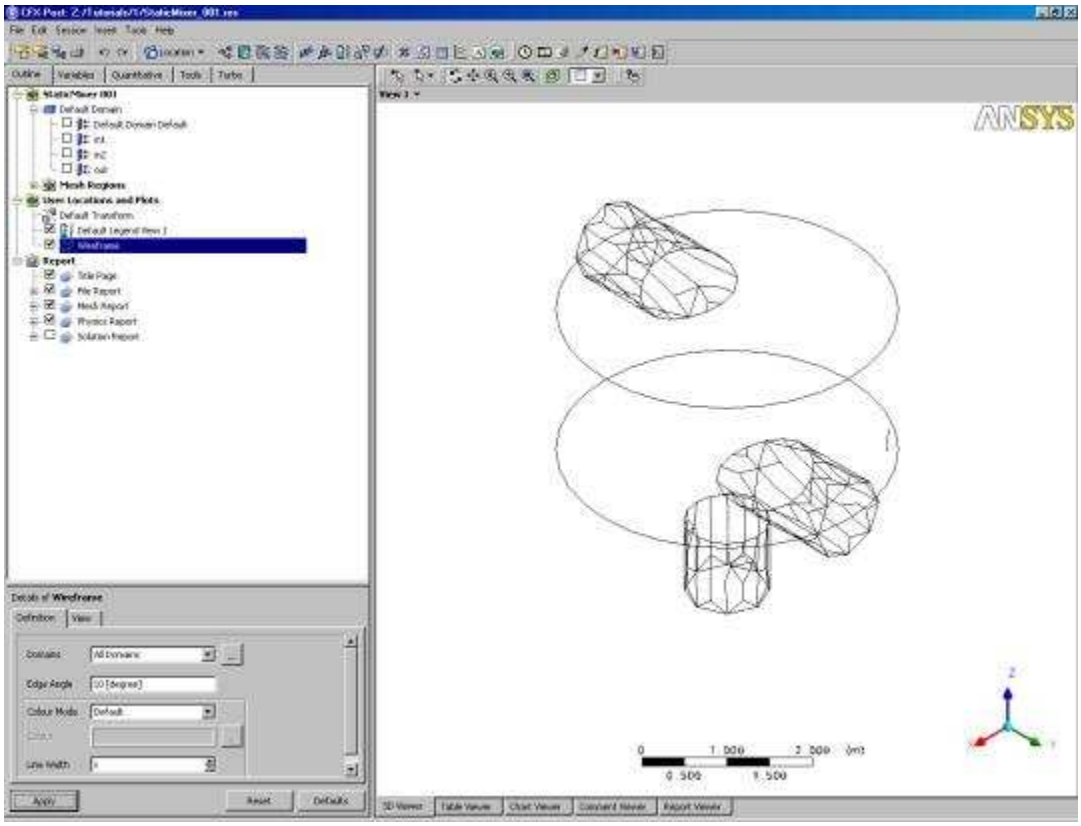
   Tip

   While it is not necessary to change the view to set the angle, do so to explore the practical uses of this feature.

2. Right-click a blank area anywhere in the viewer, select **Predefined Camera** from the shortcut menu, and select **Isometric View (Z up)**.
3. In the `Wireframe` details view, under **Definition**, click in the **Edge Angle** box.

   An embedded slider is displayed.
4. Type a value of `10 [degree]`.
5. Click **Apply** to update the object with the new setting.

   Notice that more surface mesh is displayed.



6. Drag the embedded slider to set the **Edge Angle** value to approximately `45 [degree]`.
7. Click **Apply** to update the object with the new setting.

   Less of the outline of the geometry is displayed.
8. Type a value of `15 [degree]`.
9. Click **Apply** to update the object with the new setting.

2.7.3. Creating a Point for the Origin of the Streamline

A *streamline* is the path that a particle of zero mass would follow through the domain.

1. Select **Insert** > **Location** > **Point** from the main menu.

   You can also use the toolbars to create a variety of objects. Later modules and tutorials explore this further.
2. Click **OK**.

   This accepts the default name.
3. Under **Definition**, ensure that **Method** is set to XYZ.
4. Under **Point**, enter the following coordinates: `-1, -1, 1`.

   This is a point near the first inlet.
5. Click **Apply**.

   The point appears as a symbol in the viewer as a crosshair symbol.

2.7.4. Creating a Streamline Originating from a Point

Where applicable, streamlines can trace the flow direction forwards (downstream) and/or backwards (upstream).

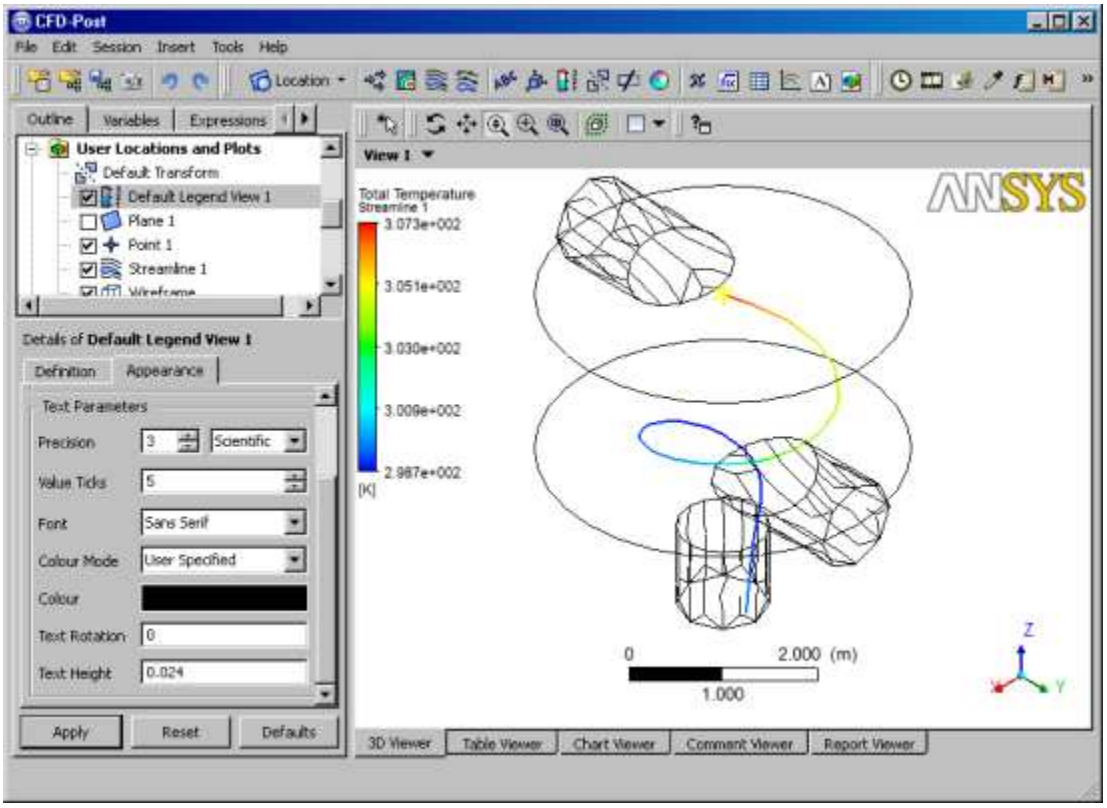
1. From the main menu, select **Insert** > **Streamline**.
2. Click **OK**.
3. Set **Definition** > **Start From** to `Point 1`.

Tip

To create streamlines originating from more than one location, click the *Ellipsis* ... icon to the right of the **Start From** box. This displays the **Location Selector** dialog box, where you can use the **Ctrl** and **Shift** keys to pick multiple locators.

4. Click the **Color** tab.
5. Set **Mode** to `Variable`.
6. Set **Variable** to `Total Temperature`.
7. Set **Range** to `Local`.
8. Click **Apply**.

The streamline shows the path of a zero mass particle from `Point 1`. The temperature is initially high near the hot inlet, but as the fluid mixes the temperature drops.



2.7.5. Rearranging the Point

Once created, a point can be rearranged manually or by setting specific coordinates.

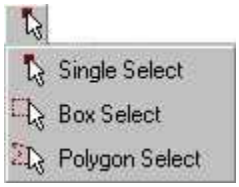
Tip

In this module, you may choose to display various views and zooms from the **Predefined Camera** option in the shortcut menu (such as **Isometric View (Z up)** or **View From -X**) and by using *Zoom Box*  if you prefer to change the display.

1. In **Outline**, under `User Locations and Plots` double-click `Point 1`.

   Properties for the selected user location are displayed.
2. Under **Point**, set these coordinates: `-1, -2.9, 1`.
3. Click **Apply**.

   The point is moved and the streamline redrawn.
4. In the viewer toolbar, click *Select*  and ensure that the adjacent toolbar icon is set to **Single Select**  .

   While in select mode, you cannot use the left mouse button to re-orient the object in the viewer.
5. In the viewer, drag `Point 1` (appears as a yellow addition sign) to a new location within the mixer.

   The point position is updated in the details view and the streamline is redrawn at the new location. The point moves normal in relation to the viewing direction.
6. Click *Rotate*  .

Tip

You can also click in the viewer area, and press the space bar to toggle between Select and Viewing Mode. A way to pick objects from Viewing Mode is to hold down **Ctrl** + **Shift** while clicking on an object with the left mouse button.

7. Under **Point**, reset these coordinates: `-1, -1, 1`.
8. Click **Apply**.

   The point appears at its original location.
9. Right-click a blank area in the viewer and select **Predefined Camera** > **View From -X**.

2.7.6. Configuring a Default Legend

You can modify the appearance of the default legend.

The default legend appears whenever a plot is created that is colored by a variable. The streamline color is based on temperature; therefore, the legend shows the temperature range. The color pattern on the legend's color bar is banded in accordance with the bands in the plot[1].

The default legend displays values for the last eligible plot that was opened in the details view. To maintain a legend definition during a CFD-Post session, you can create a new legend by clicking *Legend*  .

Because there are many settings that can be customized for the legend, this module allows you the freedom to experiment with them. In the last steps you will set up a legend, based on the default legend, with a minor modification to the position.

Tip

When editing values, you can restore the values that were present when you began editing by clicking **Reset**. To restore the factory-default values, click **Default**.

1. Double-click `Default Legend View 1`.

   The **Definition** tab of the default legend is displayed.
2. Apply the following settings

Tab	Setting	Value
Definition	Title Mode	User Specified
	Title	Streamline Temp.
	Horizontal	(Selected)
	Location > Y Justification	Bottom

3. Click **Apply**.

   The appearance and position of the legend changes based on the settings specified.
4. Modify various settings in **Definition** and click **Apply** after each change.
5. Select the **Appearance** tab.
6. Modify a variety of settings in the **Appearance** and click **Apply** after each change.
7. Click **Defaults**.
8. Click **Apply**.
9. Under **Outline**, in `User Locations and Plots`, clear the check boxes for `Point 1` and `Streamline 1`.

   Since both are no longer visible, the associated legend no longer appears.

[1] An exception occurs when one or more bands in a contour plot represent values beyond the legend's range. In this case, such bands are colored using a color that is extrapolated slightly past the range of colors shown in the legend. This can happen only when a user-specified range is used for the legend.

2.7.7. Creating a Slice Plane

Defining a slice plane allows you to obtain a cross-section of the geometry.

In CFD-Post you often view results by coloring a graphic object. The graphic object could be an isosurface, a vector plot, or in this case, a plane. The object can be a fixed color or it can vary based on the value of a variable.

You already have some objects defined by default (listed in the **Outline**). You can view results on the boundaries of the static mixer by coloring each boundary object by a variable. To view results within the geometry (that is, on non-default locators), you will create new objects.

You can use the following methods to define a plane:

- `Three Points`: creates a plane from three specified points.
- `Point and Normal`: defines a plane from one point on the plane and a normal vector to the plane.
- `YZ Plane`, `ZX Plane`, and `XY Plane`: similar to `Point and Normal`, except that the normal is defined to be normal to the indicated plane.

1. From the main menu, select **Insert** > **Location** > **Plane** or click **Location** > **Plane**.
2. In the **Insert Plane** window, type: `Slice`
3. Click **OK**.

   The **Geometry**, **Color**, **Render**, and **View** tabs let you switch between settings.
4. Click the **Geometry** tab.

2.7.8. Defining Slice Plane Geometry

You need to choose the vector normal to the plane. You want the plane to lie in the x-y plane, hence its normal vector points along the z-axis. You can specify any vector that points in the z-direction, but you will choose the most obvious (0,0,1).

1. If required, under **Geometry**, expand **Definition**.
2. Under **Method** select `Point and Normal`.
3. Under **Point** enter `0,0,1`.
4. Under **Normal** enter `0,0,1`.
5. Ensure that the **Plane Type** > **Slice** is selected.
6. Click **Apply**.

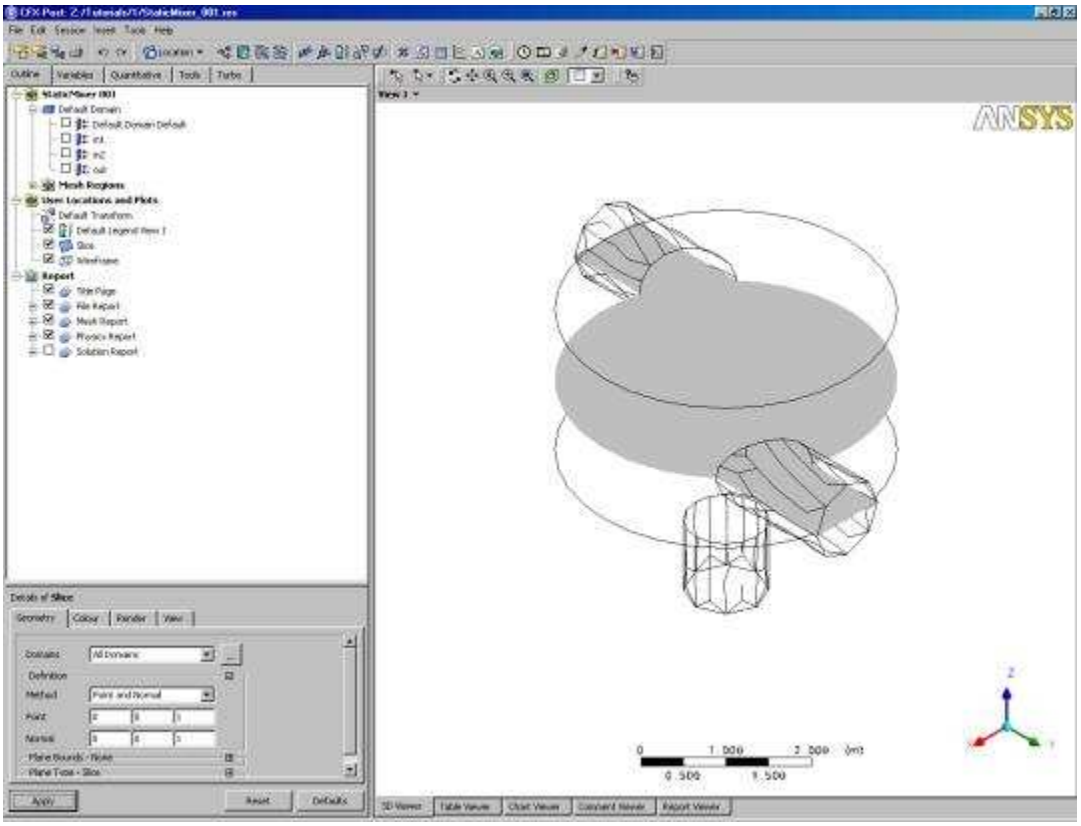
   `Slice` appears under **User Locations and Plots**. Rotate the view to see the plane.

2.7.9. Configuring Slice Plane Views

Depending on the view of the geometry, various objects may not appear because they fall in a 2D space that cannot be seen.

1. Right-click a blank area in the viewer and select **Predefined Camera** > **Isometric View (Z up)**.

   The slice is now visible in the viewer.



2. Click *Zoom Box*  .
3. Click and drag a rectangular selection over the geometry.
4. Release the mouse button to zoom in on the selection.
5. Click *Rotate*  .
6. Click and drag the mouse pointer down slightly to rotate the geometry towards you.
7. Select **Isometric View (Z up)** as described earlier.

2.7.10. Rendering Slice Planes

Render settings determine how the plane is drawn.

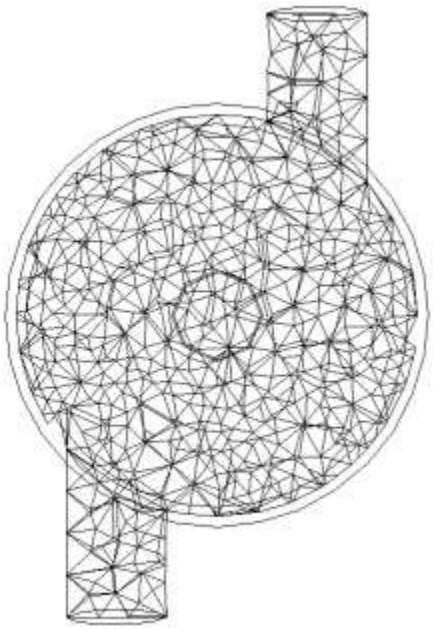
1. In the details view for **Slice**, select the **Render** tab.
2. Clear **Show Faces**.
3. Select **Show Mesh Lines**.
4. Under **Show Mesh Lines** change **Color Mode** to `User Specified`.
5. Click the current color in **Line Color** to change to a different color.

   For a greater selection of colors, click the *Ellipsis*  icon to use the **Select color** dialog box.
6. Click **Apply**.
7. Click *Zoom Box*  .
8. Zoom in on the geometry to view it in greater detail.

The line segments show where the slice plane intersects with mesh element faces. The end points of each line segment are located where the plane intersects mesh element edges.

9. Right-click a blank area in the viewer and select **Predefined Camera** > **View From +Z**.

   The image shown below can be used for comparison with *Flow in a Static Mixer (Refined Mesh)* (p. 63) (in the section *Creating a Slice Plane* (p. 71)), where a refined mesh is used.



2.7.11. Coloring the Slice Plane

The **Color** panel is used to determine how the object faces are colored.

1. Apply the following settings to `Slice`

Tab	Setting	Value
Color	Mode	Variable[a]
	Variable	Temperature
Render	Show Faces	(Selected)
	Show Mesh Lines	(Cleared)

[a]You can specify the variable (in this case, temperature) used to color the graphic element. The `Constant` mode allows you to color the plane with a fixed color.

2. Click **Apply**.

   Hot water (red) enters from one inlet and cold water (blue) from the other.

2.7.12. Moving the Slice Plane

The plane can be moved to different locations.

1. Right-click a blank area in the viewer and select **Predefined Camera > Isometric View (Z up)** from the shortcut menu.
2. Click the **Geometry** tab.

   Review the settings in **Definition** under **Point** and under **Normal**.
3. Click *Single Select*  .
4. Click and drag the plane to a new location that intersects the domain.

   As you drag the mouse, the viewer updates automatically. Note that **Point** updates with new settings.
5. Set **Point** settings to `0,0,1`.
6. Click **Apply**.
7. Click *Rotate*  .
8. Turn off visibility of `Slice` by clearing the check box next to `Slice` in the **Outline** tree view.

2.7.13. Adding Contours

Contours connect all points of equal value for a scalar variable (for example, `Temperature`) and help to visualize variable values and gradients. Colored bands fill the spaces between contour lines. Each band is colored by the average color of its two bounding contour lines (even if the latter are not displayed).

1. Right-click a blank area in the viewer and select **Predefined Camera > Isometric View (Z up)** from the shortcut menu.
2. Select **Insert** > **Contour** from the main menu or click *Contour*  .

   The **Insert Contour** dialog box is displayed.
3. Set **Name** to `Slice Contour`.
4. Click **OK**.
5. Apply the following settings

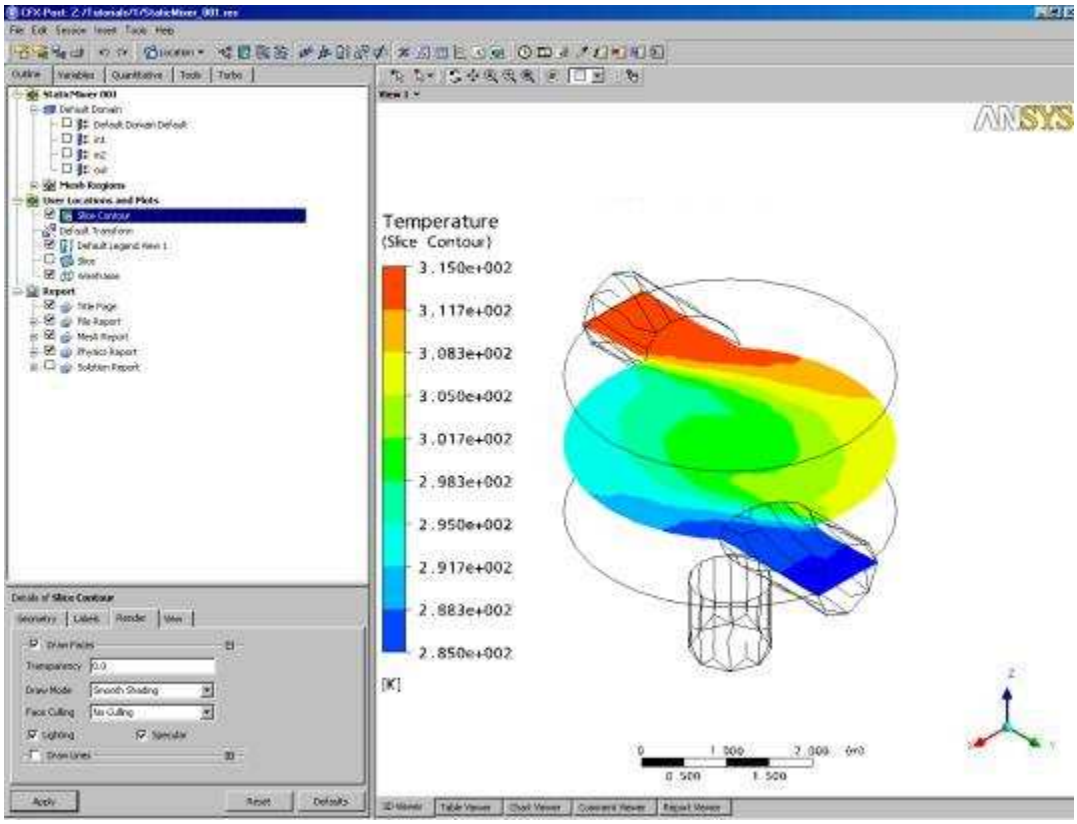
Tab	Setting	Value
Geometry	Locations	Slice
	Variable	Temperature
Render	Show Contour Bands	(Selected)

6. Click **Apply**.

   Important

   The colors of 3D graphics object faces are slightly altered when lighting is on. To view colors with highest accuracy, go to the **Render** tab and, under **Show Contour Bands**, clear **Lighting** and click **Apply**.

   The graphic element faces are visible, producing a contour plot as shown.



Note

Make sure that the visibility for `Slice` (in the **Outline** tree view) is turned off.

2.7.14. Working with Animations

Animations build transitions between views for development of video files.

2.7.14.1. Workflow Overview

This tutorial follows the general workflow for creating a keyframe animation:

1. *Showing the Animation Dialog Box* (p. 28)
2. *Creating the First Keyframe* (p. 29)
3. *Creating the Second Keyframe* (p. 30)
4. *Viewing the Animation* (p. 32)
5. *Modifying the Animation* (p. 32)
6. *Saving a Movie* (p. 33)

2.7.15. Showing the Animation Dialog Box

The **Animation** dialog box is used to define keyframes and to export to a video file.

- Select **Tools** > **Animation** or click *Animation* .

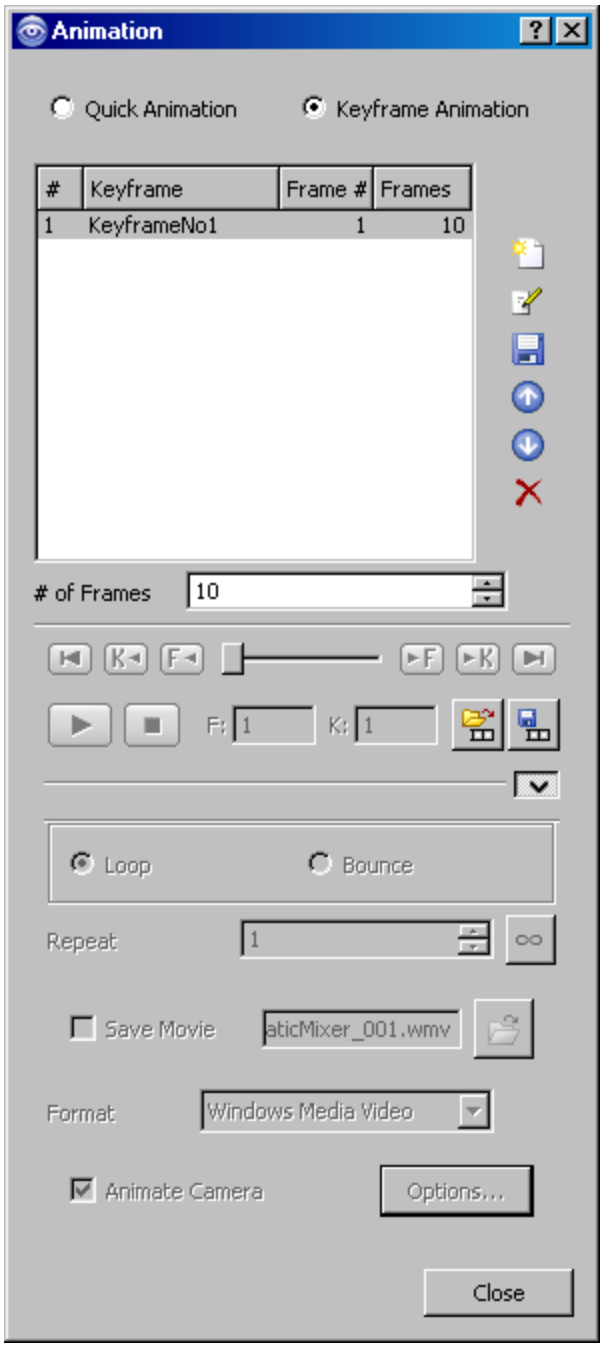
The **Animation** dialog box can be repositioned as required.

2.7.16. Creating the First Keyframe

Keyframes are required in order to produce an animation. You need to define the first viewer state, a second (and final) viewer state, and set the number of interpolated intermediate frames.

1. Right-click a blank area in the viewer and select **Predefined Camera** > **Isometric View (Z up)**.
2. In the **Outline**, under `User Locations and Plots`, turn off the visibility of `Slice Contour` and turn on the visibility of `Slice`.
3. Select the **Keyframe Animation** toggle.
4. In the **Animation** dialog box, click *New*  .

   A new keyframe named `KeyframeNo1` is created. This represents the current image displayed in the viewer.



2.7.17. Creating the Second Keyframe

Define the second keyframe and the number of intermediate frames:

1. In the **Outline**, under `User Locations and Plots`, double-click `Slice`.
2. On the **Geometry** tab, set **Point** coordinate values to `(0,0,-1.99)`.
3. Click **Apply**.

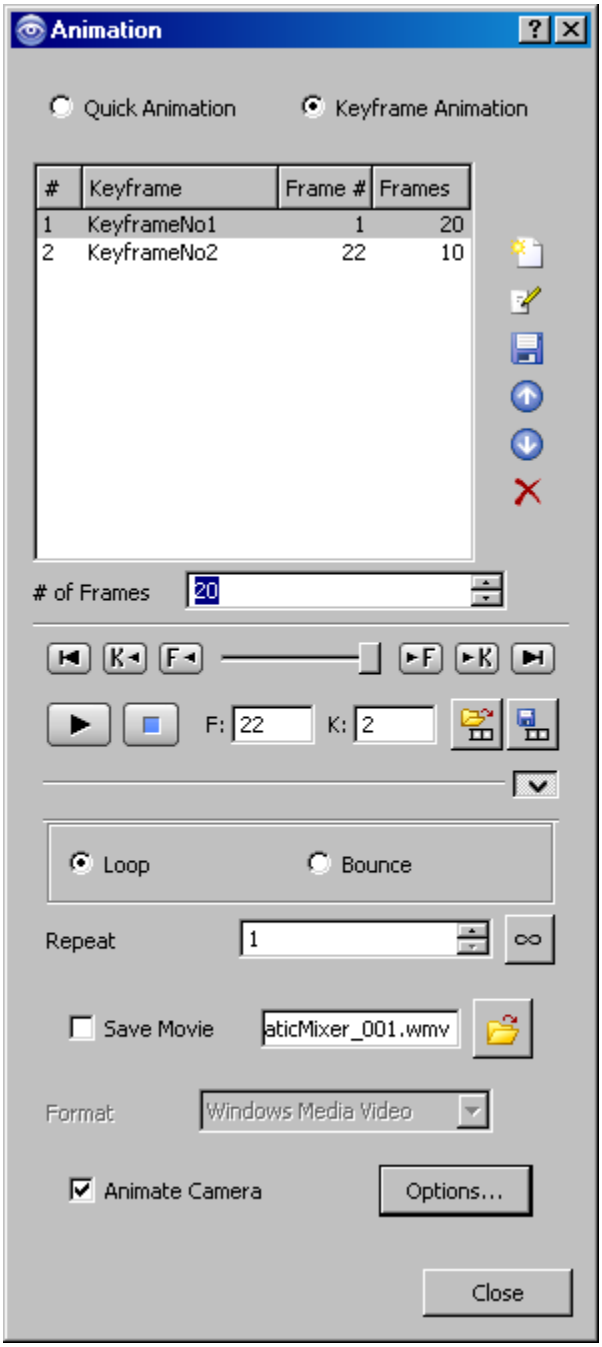
   The slice plane moves to the bottom of the mixer.
4. In the **Animation** dialog box, click *New* .

`KeyframeNo2` is created and represents the image displayed in the viewer.

5. Select `KeyframeNo1`.
6. Set **# of Frames** (located below the list of keyframes) to `20`.

   This is the number of intermediate frames used when going from `KeyframeNo1` to `KeyframeNo2`. This number is displayed in the **Frames** column for `KeyframeNo1`.
7. Press **Enter**.

   The **Frame #** column shows the frame in which each keyframe appears. `KeyframeNo1` appears at frame 1 since it defines the start of the animation. `KeyframeNo2` is at frame 22 since you have 20 intermediate frames (frames 2 to 21) in between `KeyframeNo1` and `KeyframeNo2`.



2.7.18. Viewing the Animation

More keyframes could be added, but this animation has only two keyframes (which is the minimum possible).

The controls previously greyed-out in the **Animation** dialog box are now available. The number of intermediate frames between keyframes is listed beside the keyframe having the lowest number of the pair. The number of frames listed beside the last keyframe is ignored.

1. Click *To Beginning*  .

   This ensures that the animation will begin at the first keyframe.

2. Click *Play the animation*  .

   The animation plays from frame 1 to frame 22. It plays relatively slowly because the slice plane must be updated for each frame.

2.7.19. Modifying the Animation

To make the plane sweep through the whole geometry, you will set the starting position of the plane to be at the top of the mixer. You will also modify the **Range** properties of the plane so that it shows the temperature variation better. As the animation is played, you can see the hot and cold water entering the mixer. Near the bottom of the mixer (where the water flows out) you can see that the temperature is quite uniform. The new temperature range lets you view the mixing process more accurately than the global range used in the first animation.

1. Apply the following settings to `Slice`

Tab	Setting	Value
Geometry	Point	0, 0, 1.99
Color	Mode	Variable
	Variable	Temperature
	Range	User Specified
	Min	295 [K]
	Max	305 [K]

2. Click **Apply**.

   The slice plane moves to the top of the static mixer.

   Note

   Do not double-click in the next step.

3. In the **Animation** dialog box, single click (*do not double-click*) `KeyframeNo1` to select it.

   If you had double-clicked `KeyFrameNo1`, the plane and viewer states would have been redefined according to the stored settings for `KeyFrameNo1`. If this happens, click *Undo*  and try again to select the keyframe.

4. Click *Set Keyframe*  .

   The image in the viewer replaces the one previously associated with `KeyframeNo1`.

5. Double-click `KeyframeNo2.`

   The object properties for the slice plane are updated according to the settings in `KeyFrameNo2`.

6. Apply the following settings to `Slice`

Tab	Setting	Value
Color	Mode	Variable
	Variable	Temperature
	Range	User Specified
	Min	295 [K]
	Max	305 [K]

7. Click **Apply**.
8. In the **Animation** dialog box, single-click `KeyframeNo2`.
9. Click *Set Keyframe*  to save the new settings to `KeyframeNo2`.

2.7.20. Saving a Movie

1. Click *More Animation Options*  to view the additional options.

   The **Loop** and **Bounce** radio buttons determine what happens when the animation reaches the last keyframe. When **Loop** is selected, the animation repeats itself the number of times defined by **Repeat**. When **Bounce** is selected, every other cycle is played in reverse order, starting with the second.

2. Select the check box next to **Save Movie**.
3. Set **Format** to `MPEG1`.
4. Click *Browse*  next to **Save Movie**.
5. Under **File name** type: `StaticMixer.mpg`
6. If required, set the path location to a different directory.
7. Click **Save**.

   The movie file name (including path) has been set, but the animation has not yet been produced.

8. Click *To Beginning*  .
9. Click *Play the animation*  .
10. If prompted to overwrite an existing movie click **Overwrite**.

    The animation plays and builds an MPEG file.

11. Click the **Options** button at the bottom of the **Animation** dialog box.

In **Advanced**, you can see that a **Frame Rate** of 24 frames per second was used to create the animation. The animation you produced contains a total of 22 frames, so it takes just under 1 second to play in a media player.

12. Click **Cancel** to close the dialog box.
13. Close the **Animation** dialog box.
14. Review the animation in third-party software as required.

2.7.21. Quitting CFD-Post

When finished with CFD-Post, exit the current window:

1. When you are finished, select **File** > **Quit** to exit CFD-Post.
2. Click **Quit** if prompted to save.

Chapter 3: Simulating Flow in a Static Mixer Using Workbench

This tutorial simulates a static mixer consisting of two inlet pipes delivering water into a mixing vessel; the water exits through an outlet pipe. A general workflow is established for analyzing the flow of fluid into and out of a mixer using ANSYS Workbench.

This tutorial includes:

For introductory information about ANSYS Workbench, see ANSYS CFX in ANSYS Workbench in the *CFX Introduction*.

3.1. Tutorial Features

In this tutorial you will learn about:

- Using ANSYS Workbench to set up a project.
- Using Quick Setup mode in CFX-Pre to set up a problem.
- Using ANSYS CFX-Solver Manager to obtain a solution.
- Modifying the outline plot in CFD-Post.
- Using streamlines in CFD-Post to trace the flow field from a point.
- Viewing temperature using colored planes and contours in CFD-Post.
- Creating an animation and saving it as a movie file.

Component	Feature	Details
CFX-Pre	User Mode	Quick Setup Wizard
	Analysis Type	Steady State
	Fluid Type	General Fluid
	Domain Type	Single Domain
	Turbulence Model	k-Epsilon
	Heat Transfer	Thermal Energy
	Boundary Conditions	Inlet (Subsonic)
		Outlet (Subsonic)
		Wall: No-Slip
		Wall: Adiabatic

Component	Feature	Details
	Timestep	Physical Time Scale
CFD-Post	Animation	Keyframe
	Plots	Contour
		Outline Plot (Wireframe)
		Point
		Slice Plane
		Streamline

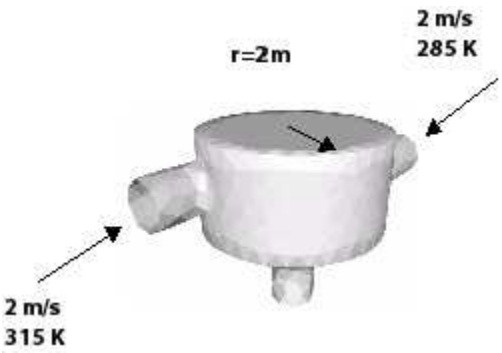
3.2. Overview of the Problem to Solve

This tutorial simulates a static mixer consisting of two inlet pipes delivering water into a mixing vessel; the water exits through an outlet pipe. A general workflow is established for analyzing the flow of fluid into and out of a mixer.

Water enters through both pipes at the same rate but at different temperatures. The first entry is at a rate of 2 m/s and a temperature of 315 K and the second entry is at a rate of 2 m/s at a temperature of 285 K. The radius of the mixer is 2 m.

Your goal in this tutorial is to understand how to use CFX in Workbench to determine the speed and temperature of the water when it exits the static mixer.

Figure 3.1 Static Mixer with 2 Inlet Pipes and 1 Outlet Pipe



3.3. Before You Begin

It is important to do the following before beginning the tutorial:

1. Use your operating system's tools to create a directory for your project's files. The directory you create will be referred to as the *working directory*.
2. Copy `StaticMixerMesh.gtm` from the `<CFXROOT>/examples` directory to the working directory, where <CFXROOT> is the installation directory for ANSYS CFX.

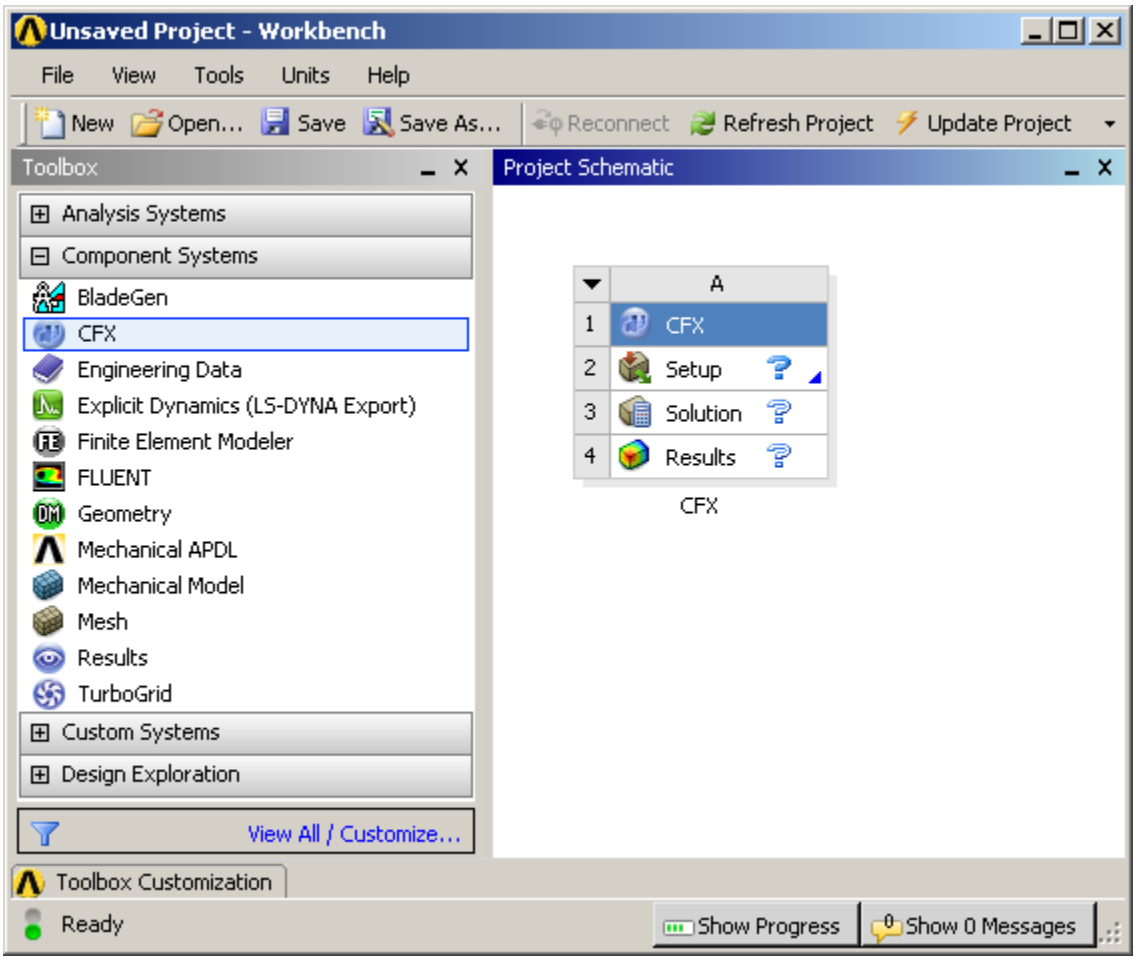
3.4. Starting ANSYS Workbench

1. Start ANSYS Workbench.

To launch ANSYS Workbench on Windows, click the **Start** menu, then select **All Programs > ANSYS 13.0 > Workbench**. To launch ANSYS Workbench on Linux, open a command line interface, type the path to "runwb2" (for example, "~/ansys_inc/v130/Framework/bin/Linux64/runwb2"), then press **Enter**.

2. In the **Toolbox** pane, open **Component Systems** and double-click **CFX**.

   A **CFX** system opens in the **Project Schematic**.



Note

You use a **CFX** component system because you are starting with a mesh. If you wanted to create the geometry and mesh, you would start with a **Fluid Flow (CFX)** analysis system.

3. From the tool bar, click **Save As** . In the dialog that appears, browse to the working directory, give the **File name** as `StaticMixer`, and click **Save**.
4. Right-click on the blue **CFX** cell (A1) and select **Rename**. Change the name of the system to `Static Mixer`.

3.5. Starting CFX-Pre from ANSYS Workbench

Because you are starting with an existing mesh, you can immediately use CFX-Pre to define the simulation. To launch CFX-Pre:

1. In ANSYS Workbench, enable **View** > **Files** and **View** > **Progress** so that you can see the files that are written and the time remaining to complete operations.
2. In the Workbench **Project Schematic**, double-click the **Setup** cell of the CFX component system. CFX-Pre opens.
3. Optionally, change the background color of the viewer in CFX-Pre for improved viewing:
   a. Select **Edit** > **Options**. The **Options** dialog box appears.
   b. Adjust the color settings under **CFX-Pre** > **Graphics Style**. For example, you could set the **Background** > **Color Type** to **Solid** and the **Color** to white.
   c. Click **OK**.

3.5.1. Creating the Simulation Definition

Before importing and working with the mesh, you need to create a simulation; in this example, you will use Quick Setup mode. Quick Setup mode provides a simple wizard-like interface for setting up simple cases. This is useful for getting familiar with the basic elements of a CFD problem setup.

1. In CFX-Pre, select **Tools** > **Quick Setup Mode**. The Quick Setup Wizard opens, enabling you to define this single-phase simulation.
2. Under **Working Fluid** select `Water`. This is a fluid already defined in the library of materials as water at 25°C.
3. Under **Mesh File**, click *Browse*  .

   The **Import Mesh** dialog box appears.
4. From your working directory, select `StaticMixerMesh.gtm` and click **Open**.

   Note

   You may have to select the `CFX Mesh (*gtm *cfx)` option under **Files of type**.

   The mesh loads, which enables you to apply physics.
5. Click **Next**.

3.5.2. Setting the Physics Definition

You need to define the type of flow and the physical models to use in the fluid domain.

The flow is steady state and you will specify the turbulence and heat transfer. Turbulence is modeled using the k-ε turbulence model and heat transfer using the thermal energy model. The k-ε turbulence model is a commonly used model and is suitable for a wide range of applications. The thermal energy model neglects high speed energy effects and is therefore suitable for low speed flow applications.

1. Under **Model Data**, note that the **Reference Pressure** is set to `1 [atm]`.

   All other pressure settings are relative to this reference pressure.
2. Set **Heat Transfer** to `Thermal Energy`.
3. Leave **Turbulence** at its default setting, `k-Epsilon`.

4. Click **Next**.

3.5.3. Defining Boundaries

The CFD model requires the definition of conditions on the boundaries of the domain.

1. Delete `Inlet` and `Outlet` from the list by right-clicking each and selecting **Delete Boundary**.
2. Right-click in the blank area where `Inlet` and `Outlet` were listed, then select **Add Boundary**.
3. Set **Name** to `in1`.
4. Click **OK**.

   The boundary is created and, when selected, properties related to the boundary are displayed.

3.5.4. Setting Boundary Data

Once boundaries are created, you need to create associated data. Based on *Figure 3.1* (p. 36), you will define the velocity and temperature for the first inlet.

1. In the **Boundary Definition** for in1, set **Boundary Type** to `Inlet`.
2. Set **Location** to `in1`.
3. Set the **Flow Specification Option** to `Normal Speed` and set **Normal Speed** to: `2 [m s^-1]`
4. Set the **Temperature Specification Static Temperature** to `315 [K]` (note the units).

3.5.5. Creating the Second Inlet Boundary Definition

Based on *Figure 3.1* (p. 36), you know the second inlet boundary condition consists of a velocity of 2 m/s and a temperature of 285 K at one of the side inlets. You will define that now.

1. Under **Boundary Definition**, right-click in the selector area and select **Add Boundary**.
2. Create a new boundary named `in2` with these settings:

Tab	Setting	Value
Boundary Data	Boundary Type	Inlet
	Location	in2
Flow Specification	Option	Normal Speed
	Normal Speed	2 [m s^-1]
Temperature Specification	Static Temperature	285 [K]

3.5.6. Creating the Outlet Boundary Definition

Now that the second inlet boundary has been created, the same concepts can be applied to building the outlet boundary.

1. Create a new boundary named `out` with these settings:

Tab	Setting	Value
Boundary Data	Boundary Type	Outlet

Tab	Setting	Value
	Location	out
Flow Specification	Option	Average Static Pressure
	Relative Pressure	0 [Pa]

2. Click **Next**.

3.5.7. Moving to General Mode

There are no further boundary conditions that need to be set. All 2D exterior regions that have not been assigned to a boundary condition are automatically assigned to the default boundary condition.

1. Set **Operation** to `Enter General Mode`.
2. Click **Finish**.

   The three boundary conditions are displayed in the viewer as sets of arrows at the boundary surfaces. Inlet boundary arrows are directed into the domain. Outlet boundary arrows are directed out of the domain.

3.5.8. Using the Viewer

Now that the simulation is loaded, take a moment to explore how you can use the viewer toolbar to zoom in or out and to rotate the object in the viewer.

3.5.8.1. Using the Zoom Tools

There are several icons available for controlling the level of zoom in the viewer.

1. Click *Zoom Box* 
2. Click and drag a rectangular box over the geometry.
3. Release the mouse button to zoom in on the selection.

   The geometry zoom changes to display the selection at a greater resolution.
4. Click *Fit View*  to re-center and re-scale the geometry.

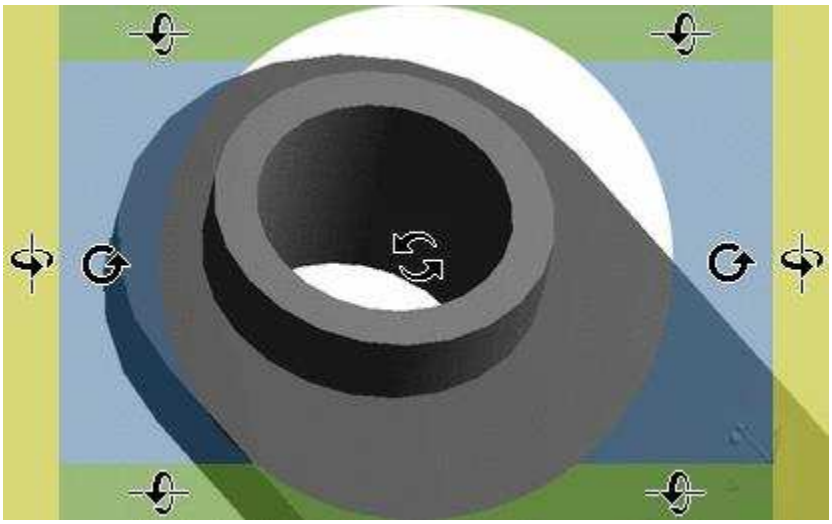
3.5.8.2. Rotating the Geometry

If you need to rotate an object or to view it from a new angle, you can use the viewer toolbar.

1. Click *Rotate*  on the viewer toolbar.
2. Click and drag within the geometry repeatedly to test the rotation of the geometry.

   The geometry rotates based on the direction of movement.

   Notice how the mouse cursor changes depending on where you are in the viewer:



3. Right-click a blank area in the viewer and select **Predefined Camera** > **View From -X**.
4. Right-click a blank area in the viewer and select **Predefined Camera** > **Isometric View (Z Up)**.

   A clearer view of the mesh is displayed.

3.5.9. Setting Solver Control

Solver Control parameters control aspects of the numerical solution generation process.

While an upwind advection scheme is less accurate than other advection schemes, it is also more robust. This advection scheme is suitable for obtaining an initial set of results, but in general should not be used to obtain final accurate results.

The time scale can be calculated automatically by the solver or set manually. The `Automatic` option tends to be conservative, leading to reliable, but often slow, convergence. It is often possible to accelerate convergence by applying a time scale factor or by choosing a manual value that is more aggressive than the `Automatic` option. In this tutorial, you will select a physical time scale, leading to convergence that is twice as fast as the `Automatic` option.

1. In the CFX-Pre tool bar, click *Solver Control*  .
2. On the **Basic Settings** tab, set **Advection Scheme** > **Option** to `Upwind`.
3. Set **Convergence Control** > **Fluid Timescale Control** > **Timescale Control** to `Physical Timescale` and set the physical timescale value to `2 [s]`.
4. Click **OK**.

3.6. Obtaining a Solution Using CFX-Solver Manager

To obtain a solution, you need to launch the CFX-Solver Manager and subsequently use it to start the solver:

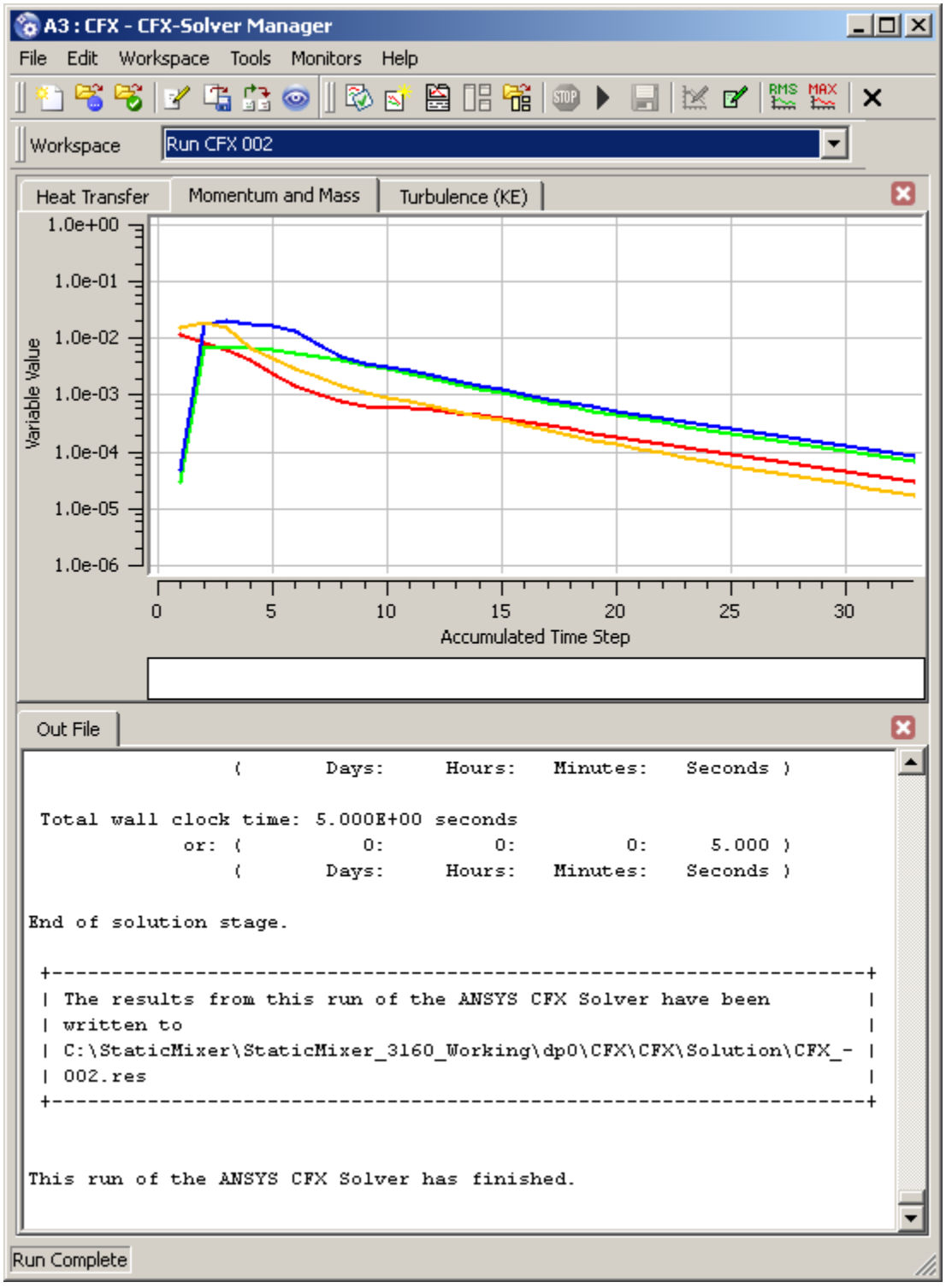
1. Double-click on the ANSYS Workbench **Solution** cell. The CFX-Solver Manager appears with the **Define Run** dialog box displayed.

   The **Define Run** dialog box enables configuration of a run for processing by CFX-Solver. In this case, all of the information required to perform a new serial run (on a single processor) is entered automatically. You do not need to alter the information in the **Define Run** dialog box.

2. Click **Start Run**.

   CFX-Solver launches and a split screen appears and displays the results of the run graphically and as text. The panes continue to build as CFX-Solver Manager operates.

   One window shows the convergence history plots and the other displays text output from CFX-Solver. The text lists physical properties, boundary conditions, and various other parameters used or calculated in creating the model. All the text is written to the output file automatically (in this case, `StaticMixer_001.out`).



Note

Once the second iteration appears, data begins to plot. Plotting may take a long time depending on the amount of data to process. Let the process run.

3.6.1. Display the Results in CFD-Post

Once CFX-Solver has finished, you can use CFD-Post to review the finished results:

1. When CFX-Solver is finished, a message is displayed and the final line in the `.out` file (which you can see in the CFX-Solver Manager) is:

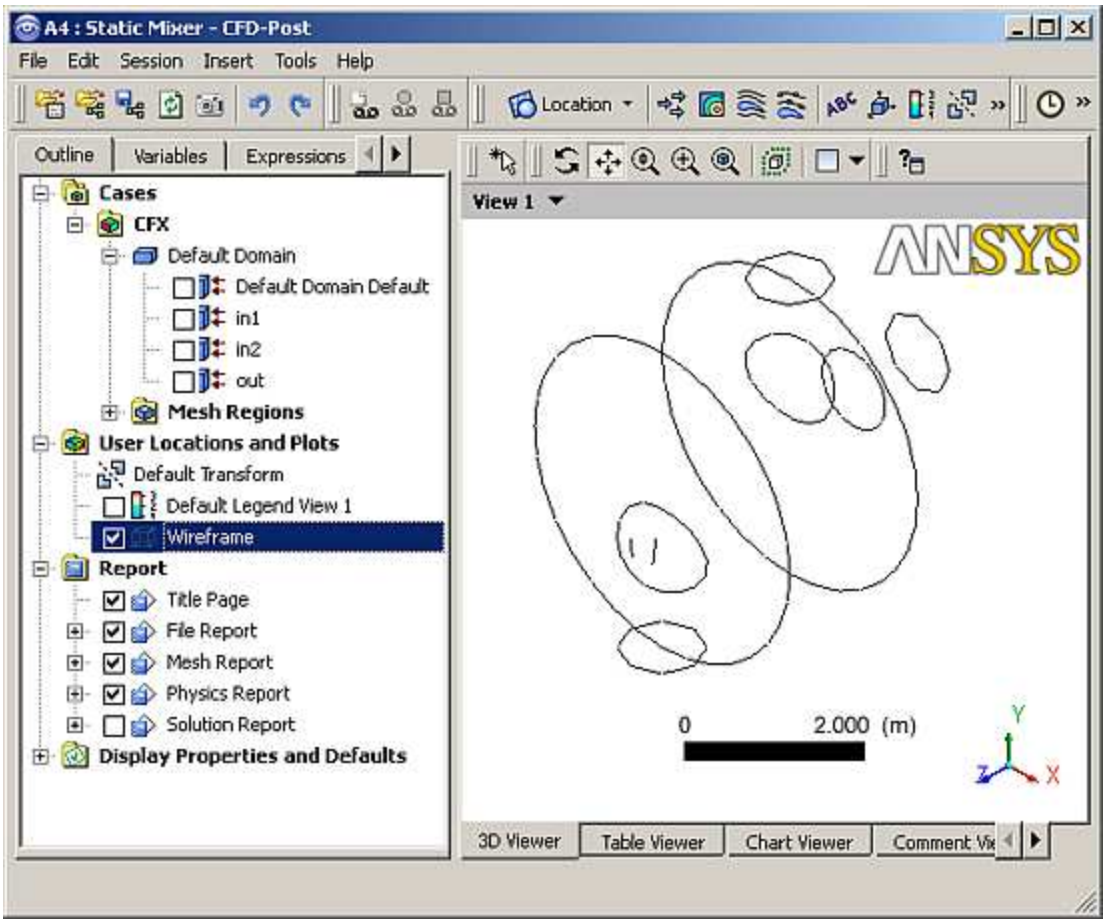
   ```
   This run of the ANSYS CFX Solver has finished.
   ```

2. In ANSYS Workbench, right-click on the **Results** cell and select **Refresh**.
3. When the refresh is complete, double-click on the **Results** cell. CFD-Post appears.

3.7. Viewing the Results in CFD-Post

When CFD-Post starts, the viewer and **Outline** workspace are displayed. Optionally, change the background color of the viewer improved viewing:

1. In CFD-Post, select **Edit** > **Options**. The **Options** dialog box appears.
2. Adjust the color settings under **CFD-Post** > **Viewer**. For example, you could set the **Background** > **Color Type** to **Solid** and the **Color** to white.
3. Click **OK**.



The viewer displays an outline of the geometry and other graphic objects. You can use the mouse or the toolbar icons to manipulate the view, exactly as in CFX-Pre.

3.7.1. Workflow Overview

This tutorial describes the following workflow for viewing results in CFD-Post:

1. *Setting the Edge Angle for a Wireframe Object* (p. 45)
2. *Creating a Point for the Origin of the Streamline* (p. 46)
3. *Creating a Streamline Originating from a Point* (p. 47)
4. *Rearranging the Point* (p. 48)
5. *Configuring a Default Legend* (p. 49)
6. *Creating a Slice Plane* (p. 50)
7. *Defining Slice Plane Geometry* (p. 51)
8. *Configuring Slice Plane Views* (p. 51)
9. *Rendering Slice Planes* (p. 52)
10. *Coloring the Slice Plane* (p. 53)
11. *Moving the Slice Plane* (p. 54)
12. *Adding Contours* (p. 54)
13. *Working with Animations* (p. 55)

3.7.2. Setting the Edge Angle for a Wireframe Object

The outline of the geometry is called the *wireframe* or *outline plot*.

By default, CFD-Post displays only some of the surface mesh. This sometimes means that when you first load your results file, the geometry outline is not displayed clearly. You can control the amount of the surface mesh shown by editing the `Wireframe` object listed in the **Outline**.

The check boxes next to each object name in the **Outline** tree view control the visibility of each object. Currently only the `Wireframe` and `Default Legend` objects have visibility turned on.

The edge angle determines how much of the surface mesh is visible. If the angle between two adjacent faces is greater than the edge angle, then that edge is drawn. If the edge angle is set to 0°, the entire surface mesh is drawn. If the edge angle is large, then only the most significant corner edges of the geometry are drawn.

For this geometry, a setting of approximately 15° lets you view the model location without displaying an excessive amount of the surface mesh.

In this module you can also modify the zoom settings and view of the wireframe.

1. In the **Outline**, under `User Locations and Plots`, double-click `Wireframe`.
2. Right-click a blank area anywhere in the viewer, select **Predefined Camera** from the shortcut menu, and select **Isometric View (Z up)**.

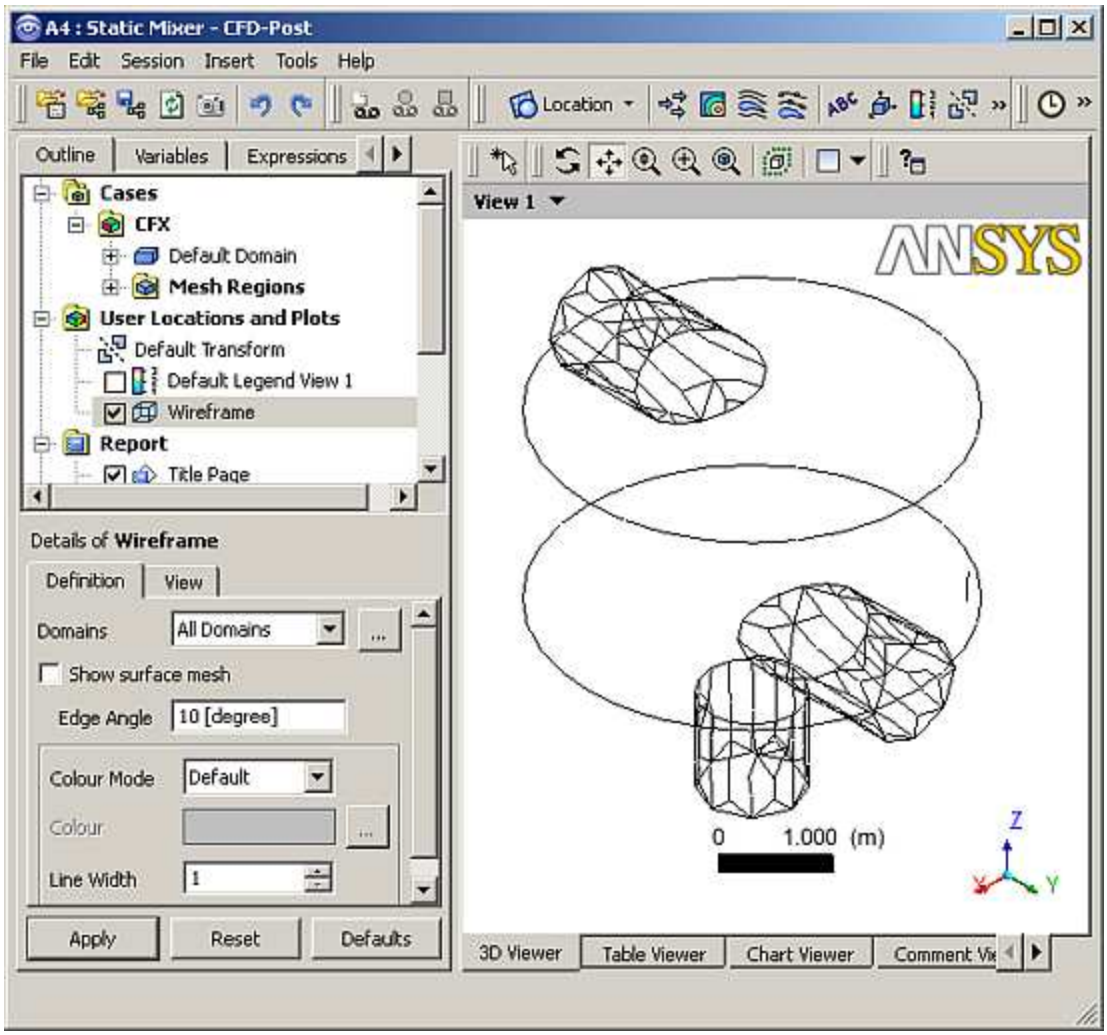
   Tip

   While it is not necessary to change the view to set the edge angle for the wireframe, doing so enables you to explore the practical uses of this feature.

3. In the `Wireframe` details view, under **Definition**, click in the **Edge Angle** box.

   An embedded slider is displayed.
4. Type a value of `10 [degree]`.
5. Click **Apply** to update the object with the new setting.

   Notice that more surface mesh is displayed.



6. Drag the embedded slider to set the **Edge Angle** value to approximately `45 [degree]`.
7. Click **Apply** to update the object with the new setting.

   Less of the outline of the geometry is displayed.
8. Type a value of `15 [degree]`.
9. Click **Apply** to update the object with the new setting.

3.7.3. Creating a Point for the Origin of the Streamline

A *streamline* is the path that a particle of zero mass would follow through the domain.

1. Select **Insert** > **Location** > **Point** from the main menu.

   You can also use the toolbars to create a variety of objects. Later modules and tutorials will explore this further.
2. Click **OK**.

   This accepts the default name.
3. Set **Definition** > **Method** to XYZ.
4. Under **Point**, enter the following coordinates: `-1, -1, 1`.

This is a point near the first inlet.

5. Click **Apply**.

   The point appears as a symbol in the viewer as a crosshair symbol.

3.7.4. Creating a Streamline Originating from a Point

Where applicable, streamlines can trace the flow direction forwards (downstream) and/or backwards (upstream).

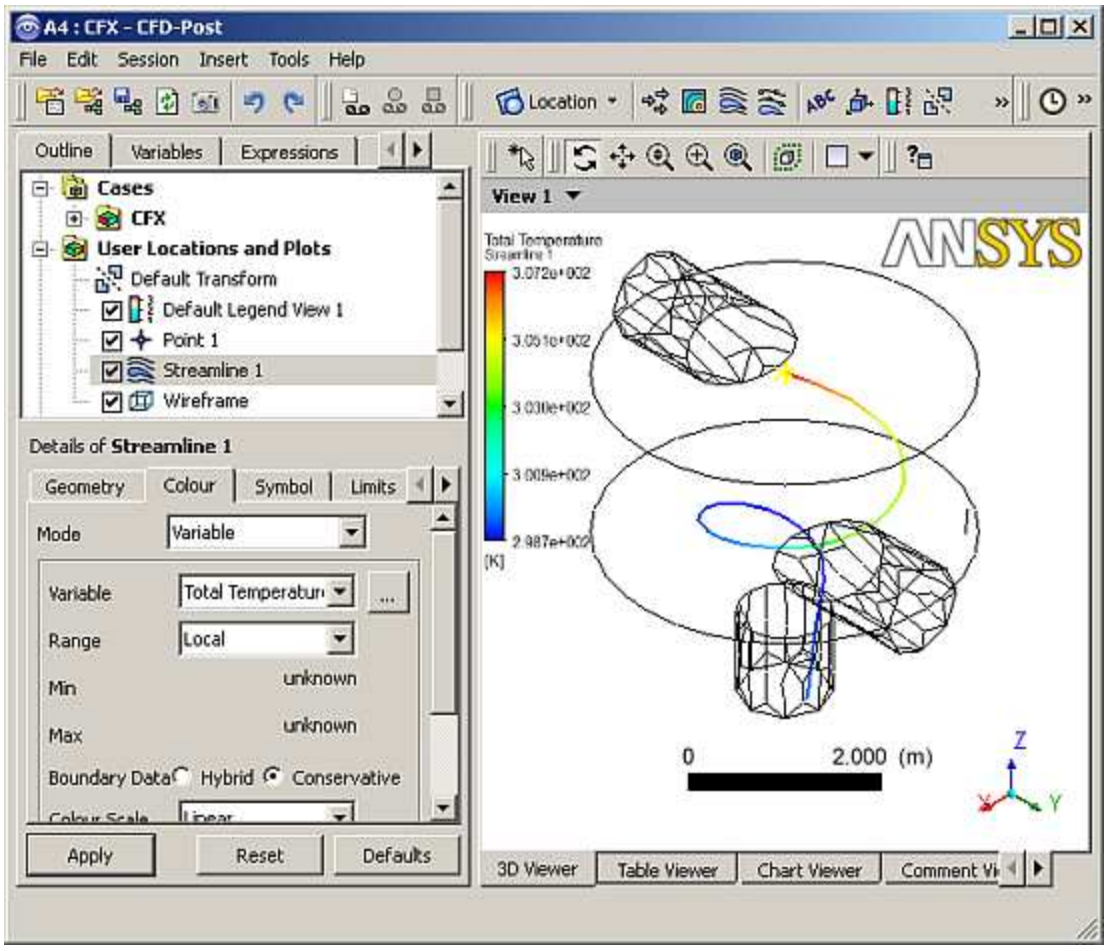
1. From the main menu, select **Insert** > **Streamline**.
2. Click **OK**.
3. Set **Definition** > **Start From** to `Point 1`.

   Tip

   To create streamlines originating from more than one location, click the *Ellipsis* ... icon to the right of the **Start From** box. This displays the **Location Selector** dialog box, where you can use the **Ctrl** and **Shift** keys to pick multiple locators.

4. Click the **Color** tab.
5. Set **Mode** to `Variable`.
6. Set **Variable** to `Total Temperature`.
7. Set **Range** to `Local`.
8. Click **Apply**.

   The streamline shows the path of a zero mass particle from `Point 1`. The temperature is initially high near the hot inlet, but as the fluid mixes the temperature drops.



3.7.5. Rearranging the Point

Once created, a point can be rearranged manually or by setting specific coordinates.

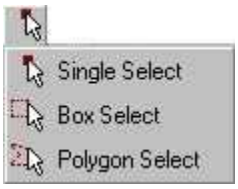
Tip

In this module, you may choose to display various views and zooms from the **Predefined Camera** option in the shortcut menu (such as **Isometric View (Z up)** or **View From -X**) and by using *Zoom Box*  if you prefer to change the display.

1. In **Outline**, under `User Locations and Plots` double-click `Point 1`.

   Properties for the selected user location are displayed.
2. Under **Point**, set these coordinates: `-1, -2.9, 1`.
3. Click **Apply**.

   The point is moved and the streamline redrawn.
4. In the viewer toolbar, click *Select*  and ensure that the adjacent toolbar icon is set to **Single Select**  .



While in select mode, you cannot use the left mouse button to re-orient the object in the viewer.

5. In the viewer, drag `Point 1` (appears as a yellow addition sign) to a new location within the mixer.

   The point position is updated in the details view and the streamline is redrawn at the new location. The point moves normal in relation to the viewing direction.

6. Click *Rotate* .

Tip

You can also click in the viewer area, and press the space bar to toggle between Select and Viewing Mode. A way to pick objects from Viewing Mode is to hold down **Ctrl** + **Shift** while clicking on an object with the left mouse button.

---

7. Under **Point**, reset these coordinates: `-1, -1, 1`.
8. Click **Apply**.

   The point appears at its original location.

9. Right-click a blank area in the viewer and select **Predefined Camera** > **View From -X**.

3.7.6. Configuring a Default Legend

You can modify the appearance of the default legend.

The default legend appears whenever a plot is created that is colored by a variable. The streamline color is based on temperature; therefore, the legend shows the temperature range. The color pattern on the legend's color bar is banded in accordance with the bands in the plot[1].

The default legend displays values for the last eligible plot that was opened in the details view. To maintain a legend definition during a CFD-Post session, you can create a new legend by clicking *Legend* .

Because there are many settings that can be customized for the legend, this module allows you the freedom to experiment with them. In the last steps you will set up a legend, based on the default legend, with a minor modification to the position.

Tip

When editing values, you can restore the values that were present when you began editing by clicking **Reset**. To restore the factory-default values, click **Default**.

---

[1] An exception occurs when one or more bands in a contour plot represent values beyond the legend's range. In this case, such bands are colored using a color that is extrapolated slightly past the range of colors shown in the legend. This can happen only when a user-specified range is used for the legend.

1. Double-click `Default Legend View 1`.

   The **Definition** tab of the default legend is displayed.
2. Apply the following settings

Tab	Setting	Value
Definition	Title Mode	User Specified
	Title	Streamline Temp.
	Horizontal	(Selected)
	Location > Y Justification	Bottom

3. Click **Apply**.

   The appearance and position of the legend changes based on the settings specified.
4. Modify various settings in **Definition** and click **Apply** after each change.
5. Select **Appearance**.
6. Modify a variety of settings in the **Appearance** and click **Apply** after each change.
7. Click **Defaults**.
8. Click **Apply**.
9. Under **Outline**, in `User Locations and Plots`, clear the check boxes for `Point 1` and `Streamline 1`.

   Since both are no longer visible, the associated legend no longer appears.

3.7.7. Creating a Slice Plane

Defining a slice plane allows you to obtain a cross-section of the geometry.

In CFD-Post you often view results by coloring a graphic object. The graphic object could be an isosurface, a vector plot, or in this case, a plane. The object can be a fixed color or it can vary based on the value of a variable.

You already have some objects defined by default (listed in the **Outline**). You can view results on the boundaries of the static mixer by coloring each boundary object by a variable. To view results within the geometry (that is, on non-default locators), you will create new objects.

You can use the following methods to define a plane:

- `Three Points`: creates a plane from three specified points.
- `Point and Normal`: defines a plane from one point on the plane and a normal vector to the plane.
- `YZ Plane`, `ZX Plane`, and `XY Plane`: similar to `Point and Normal`, except that the normal is defined to be normal to the indicated plane.

1. From the main menu, select **Insert** > **Location** > **Plane** or click **Location** > **Plane**.
2. In the **Insert Plane** window, type: `Slice`
3. Click **OK**.

The details view for the plane appears; the **Geometry**, **Color**, **Render**, and **View** tabs let you configure the characteristics of the plane.

3.7.8. Defining Slice Plane Geometry

You need to choose the vector normal to the plane. You want the plane to lie in the x-y plane, hence its normal vector points along the z-axis. You can specify any vector that points in the z-direction, but you will choose the most obvious (0,0,1).

1. On the **Geometry** tab, expand **Definition**.
2. Under **Method** select `Point and Normal`.
3. Under **Point** enter `0,0,1`.
4. Under **Normal** enter `0,0,1`.
5. Click **Apply**.

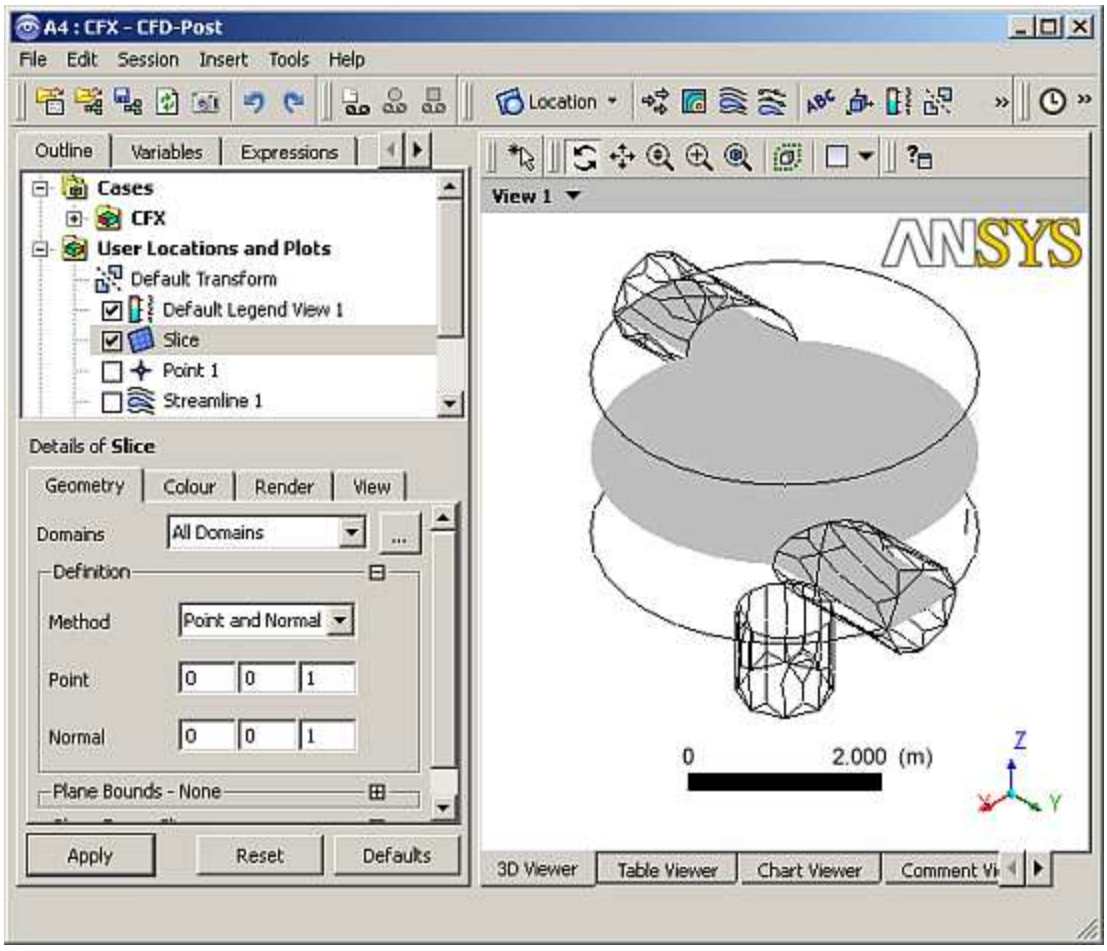
   `Slice` appears under **User Locations and Plots**. Rotate the view to see the plane.

3.7.9. Configuring Slice Plane Views

Depending on the view of the geometry, various objects may not appear because they fall in a 2D space that cannot be seen.

1. Right-click a blank area in the viewer and select **Predefined Camera** > **Isometric View (Z up)**.

   The slice is now visible in the viewer.



2. Click *Zoom Box* .
3. Click and drag a rectangular selection over the geometry.
4. Release the mouse button to zoom in on the selection.
5. Click *Rotate* .
6. Click and drag the mouse pointer down slightly to rotate the geometry towards you.
7. Select **Isometric View (Z up)** as described earlier.

3.7.10. Rendering Slice Planes

Render settings determine how the plane is drawn.

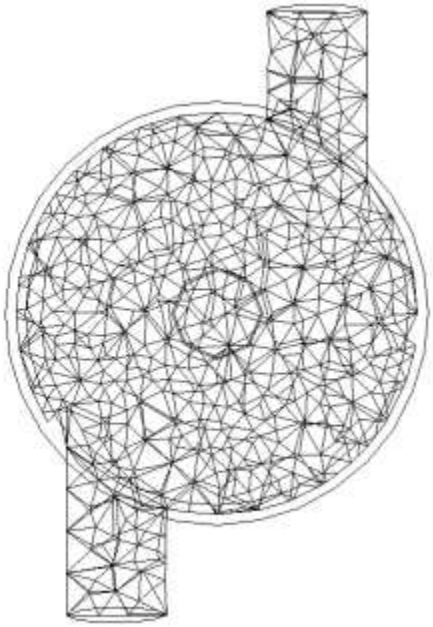
1. In the details view for **Slice**, select the **Render** tab.
2. Clear **Show Faces**.
3. Select **Show Mesh Lines**.
4. Under **Show Mesh Lines** change **Color Mode** to `User Specified`.
5. Click the current color in **Line Color** to change to a different color.

   For a greater selection of colors, click the *Ellipsis* ... icon to use the **Color selector** dialog box.
6. Click **Apply**.

7. Click *Zoom Box*  .
8. Zoom in on the geometry to view it in greater detail.

   The line segments show where the slice plane intersects with mesh element faces. The end points of each line segment are located where the plane intersects mesh element edges.
9. Right-click a blank area in the viewer and select **Predefined Camera** > **View From +Z**.

   The image shown below can be used for comparison with *Flow in a Static Mixer (Refined Mesh)* (p. 63) (in the section *Creating a Slice Plane* (p. 71)), where a refined mesh is used.



3.7.11. Coloring the Slice Plane

The **Color** panel is used to determine how the object faces are colored.

1. Apply the following settings to `Slice`

Tab	Setting	Value
Color	Mode	Variable [1 (p. 53)]
	Variable	Temperature
Render	Show Faces	(Selected)
	Show Mesh Lines	(Cleared)

Footnote

1. You can specify the variable (in this case, temperature) used to color the graphic element. The `Constant` mode allows you to color the plane with a fixed color.

2. Click **Apply**.

Hot water (red) enters from one inlet and cold water (blue) from the other.

3.7.12. Moving the Slice Plane

You can move the plane to different locations:

1. Right-click a blank area in the viewer and select **Predefined Camera** > **Isometric View (Z up)** from the shortcut menu.
2. Click the **Geometry** tab.

   Review the settings in **Definition** under **Point** and under **Normal**.
3. Click *Single Select*  .
4. Click and drag the plane to a new location that intersects the domain.

   As you drag the mouse, the viewer updates automatically. Note that **Point** updates with new settings.
5. Type in **Point** settings of `0,0,1`.
6. Click **Apply**.
7. Click *Rotate*  .
8. Turn off the visibility for `Slice` by clearing the check box next to `Slice` in the **Outline** tree view.

3.7.13. Adding Contours

Contours connect all points of equal value for a scalar variable (for example, `Temperature`) and help to visualize variable values and gradients. Colored bands fill the spaces between contour lines. Each band is colored by the average color of its two bounding contour lines (even if the latter are not displayed).

1. Right-click a blank area in the viewer and select **Predefined Camera > Isometric View (Z up)** from the shortcut menu.
2. Select **Insert** > **Contour** from the main menu or click *Contour*  .

   The **Insert Contour** dialog box is displayed.
3. Set **Name** to `Slice Contour`.
4. Click **OK**.
5. Apply the following settings

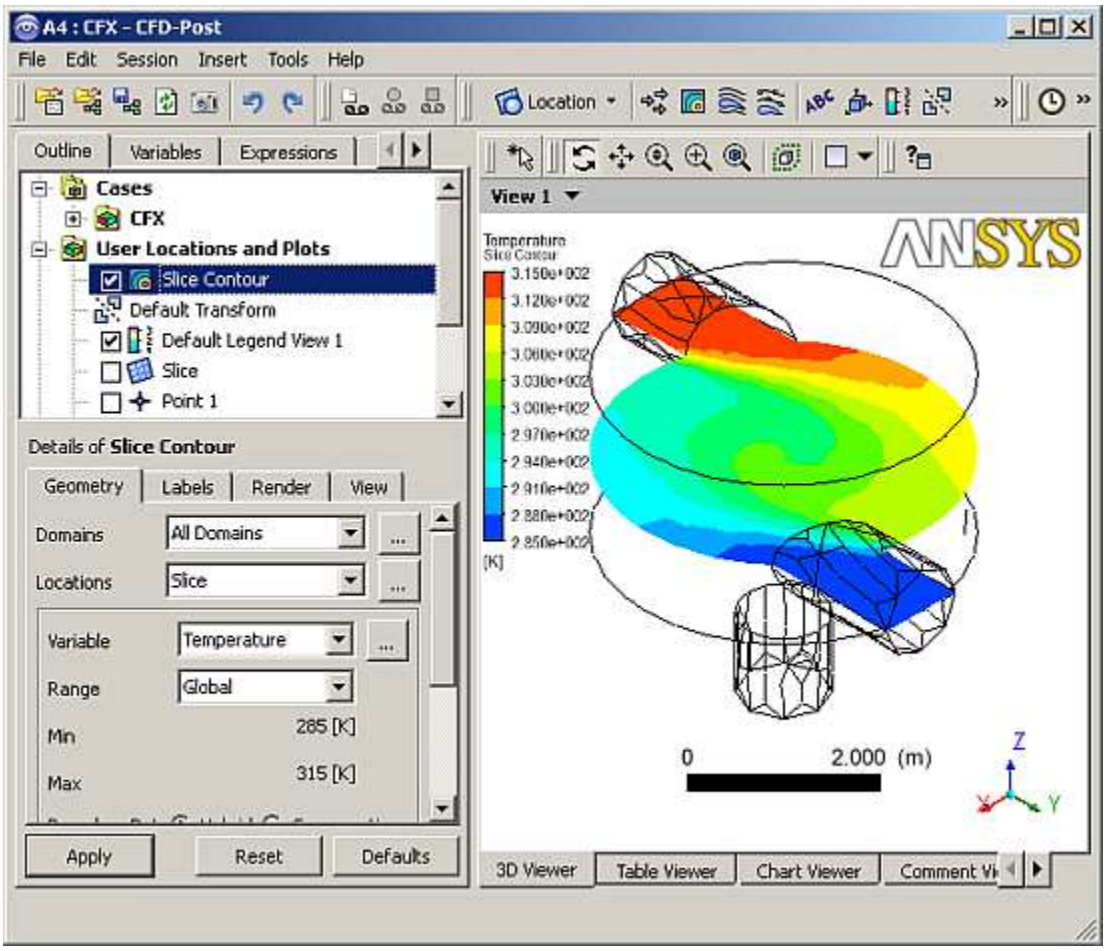
Tab	Setting	Value
Geometry	Locations	Slice
	Variable	Temperature
Render	Show Contour Lines	(Selected)

6. Click **Apply**.

Important

The colors of 3D graphics object faces are slightly altered when lighting is on. To view colors with highest accuracy, go to the **Render** tab and, under **Show Faces**, clear **Lighting** and click **Apply**.

The graphic element faces are visible, producing a contour plot as shown.



Note

Make sure that the visibility of `Slice` (in the **Outline** tree view) is turned off.

3.7.14. Working with Animations

Animations build transitions between views for development of video files.

3.7.14.1. Workflow Overview

This tutorial follows the general workflow for creating a keyframe animation:

1. *Showing the Animation Dialog Box* (p. 56)
2. *Creating the First Keyframe* (p. 56)

3. *Creating the Second Keyframe* (p. 57)
4. *Viewing the Animation* (p. 59)
5. *Modifying the Animation* (p. 59)
6. *Saving a Movie* (p. 60)

3.7.15. Showing the Animation Dialog Box

The **Animation** dialog box is used to define keyframes and to export to a video file.

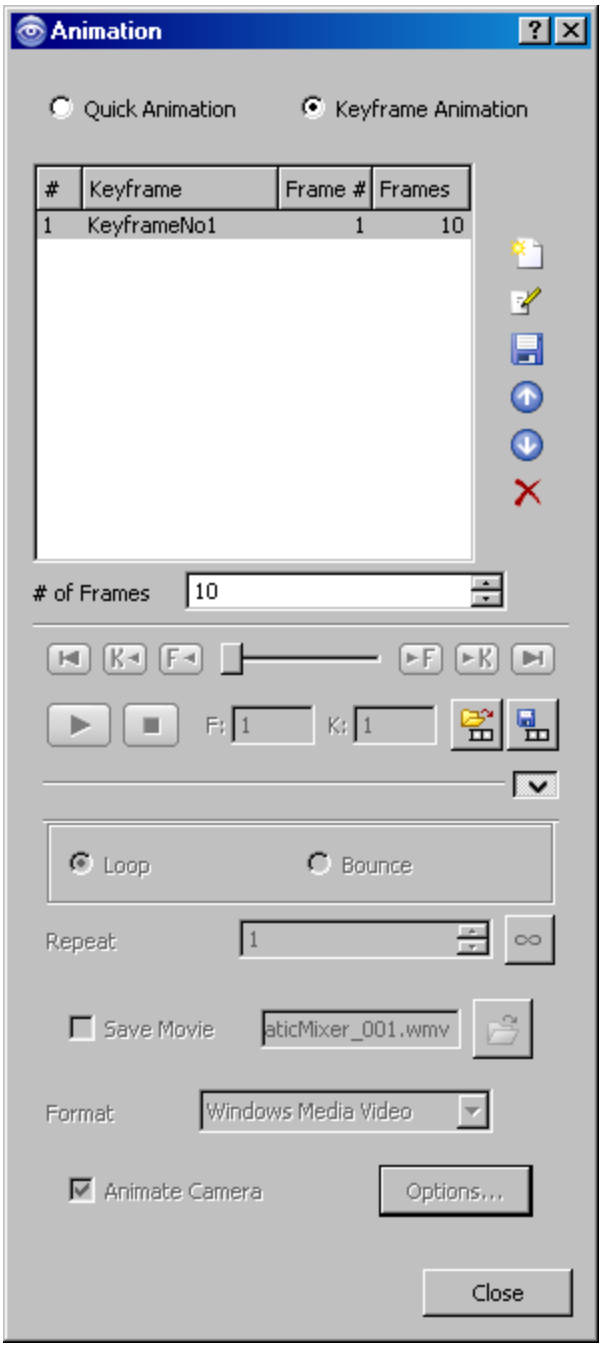
1. Select **Tools** > **Animation** or click *Animation*  .
2. Select **Keyframe Animation**.

3.7.16. Creating the First Keyframe

Keyframes are required in order to produce a keyframe animation. You need to define the first viewer state, a second (and final) viewer state, and set the number of interpolated intermediate frames.

1. Right-click a blank area in the viewer and select **Predefined Camera** > **Isometric View (Z up)**.
2. In the **Outline**, under `User Locations and Plots`, turn off the visibility of `Slice Contour` and turn on the visibility of `Slice`.
3. In the **Animation** dialog box, click *New*  .

   A new keyframe named `KeyframeNo1` is created. This represents the current image displayed in the viewer.



3.7.17. Creating the Second Keyframe

Define the second keyframe and the number of intermediate frames:

1. In the **Outline**, under `User Locations and Plots`, double-click `Slice`.
2. On the **Geometry** tab, set **Point** coordinate values to `(0,0,-1.99)`.
3. Click **Apply**.

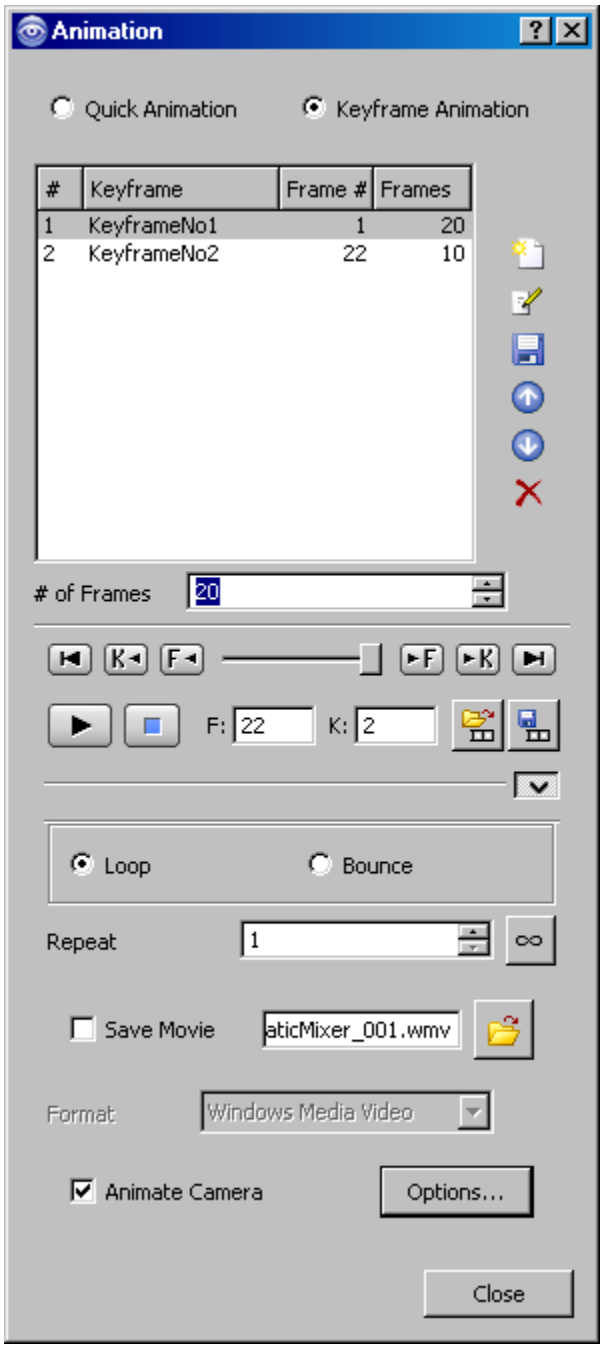
   The slice plane moves to the bottom of the mixer.
4. In the **Animation** dialog box, click *New* .

KeyframeNo2 is created and represents the image displayed in the viewer.

5. Select KeyframeNo1 so that you can set the number of frames to be interpolated between the two keyframes.
6. Set **# of Frames** (located below the list of keyframes) to 20.

   This is the number of intermediate frames used when going from KeyframeNo1 to KeyframeNo2. This number is displayed in the **Frames** column for KeyframeNo1.
7. Press **Enter**.

   The **Frame #** column shows the frame in which each keyframe appears. KeyframeNo1 appears at frame 1 since it defines the start of the animation. KeyframeNo2 is at frame 22 since you have 20 intermediate frames (frames 2 to 21) in between KeyframeNo1 and KeyframeNo2.



3.7.18. Viewing the Animation

More keyframes could be added, but this animation has only two keyframes (which is the minimum possible).

The controls previously greyed-out in the **Animation** dialog box are now available. The number of intermediate frames between keyframes is listed beside the keyframe having the lowest number of the pair. The number of frames listed beside the last keyframe is ignored.

1. Click *To Beginning*  .

   This ensures that the animation will begin at the first keyframe.

2. Click *Play the animation*  .

   The animation plays from frame 1 to frame 22. It plays relatively slowly because the slice plane must be updated for each frame.

3.7.19. Modifying the Animation

To make the plane sweep through the whole geometry, you will set the starting position of the plane to be at the top of the mixer. You will also modify the **Range** properties of the plane so that it shows the temperature variation better. As the animation is played, you can see the hot and cold water entering the mixer. Near the bottom of the mixer (where the water flows out) you can see that the temperature is quite uniform. The new temperature range lets you view the mixing process more accurately than the global range used in the first animation.

1. Apply the following settings to `Slice`

Tab	Setting	Value
Geometry	Point	0, 0, 1.99
Color	Mode	Variable
	Variable	Temperature
	Range	User Specified
	Min	295 [K]
	Max	305 [K]

2. Click **Apply**.

   The slice plane moves to the top of the static mixer.

   Note

   Do not double-click in the next step.

3. In the **Animation** dialog box, single click (*do not double-click*) `KeyframeNo1` to select it.

   If you had double-clicked `KeyFrameNo1`, the plane and viewer states would have been redefined according to the stored settings for `KeyFrameNo1`. If this happens, click *Undo*  and try again to select the keyframe.

4. Click *Set Keyframe*  .

   The image in the viewer replaces the one previously associated with `KeyframeNo1`.

5. Double-click `KeyframeNo2`.

   The object properties for the slice plane are updated according to the settings in `KeyFrameNo2`.

6. Apply the following settings to `Slice`

Tab	Setting	Value
Color	Mode	Variable
	Variable	Temperature
	Range	User Specified
	Min	295 [K]
	Max	305 [K]

7. Click **Apply**.
8. In the **Animation** dialog box, single-click `KeyframeNo2`.
9. Click *Set Keyframe*  to save the new settings to `KeyframeNo2`.

3.7.20. Saving a Movie

1. Click *More Animation Options*  to view the additional options.

   The **Loop** and **Bounce** radio buttons determine what happens when the animation reaches the last keyframe. When **Loop** is selected, the animation repeats itself the number of times defined by **Repeat**. When **Bounce** is selected, every other cycle is played in reverse order, starting with the second.

2. Select **Save Movie**.
3. Set **Format** to `MPEG1`.
4. Click *Browse*  next to **Save Movie**.
5. Under **File name** type: `StaticMixer.mpg`
6. If required, set the path location to a different directory. You may want to set the directory to your working directory so that the animation will be in the same location as the project files; however, ANSYS Workbench will not store the file as part of the project.
7. Click **Save**.

   The movie file name (including path) has been set, but the animation has not yet been produced.

8. Click *To Beginning*  .
9. Click *Play the animation*  .
10. If prompted to overwrite an existing movie click **Overwrite**.

    The animation plays and builds an MPEG file.

11. Click the **Options** button at the bottom of the **Animation** dialog box.

In **Advanced**, you can see that a **Frame Rate** of 24 frames per second was used to create the animation. The animation you produced contains a total of 22 frames, so it takes just under 1 second to play in a media player.

12. Click **Cancel** to close the dialog box.
13. Close the **Animation** dialog box.
14. View the animation using a media player.

3.7.21. Closing the Applications

Before you close the project, take a moment to look at the files listed in the **Files** view. You will see the project file, `StaticMixer.wbpj`, and the files that ANSYS Workbench created (CFX-Solver Input, CFX-Solver Output, CFX-Solver Results, CFX-Pre Case, CFD-Post State, and Design Point files). Any files that you added to that part of the file system (such as the movie file) will not be listed.

Close ANSYS Workbench (and the applications it launched) by selecting **File** > **Exit** from ANSYS Workbench. ANSYS Workbench prompts you to save all of your project files.

Chapter 4: Flow in a Static Mixer (Refined Mesh)

This tutorial includes:

4.1. Tutorial Features

In this tutorial you will learn about:

- Using the General Mode of CFX-Pre (this mode is used for more complex cases).
- Rerunning a problem with a refined mesh.
- Importing a CCL (CFX Command Language) file to copy the definition of a different simulation into the current simulation.
- Viewing the mesh with a Sphere Volume locator and a Surface Plot.
- Using the Report Viewer to analyze mesh quality.

Component	Feature	Details
CFX-Pre	User Mode	General Mode
	Analysis Type	Steady State
	Fluid Type	General Fluid
	Domain Type	Single Domain
	Turbulence Model	k-Epsilon
	Heat Transfer	Thermal Energy
	Boundary Conditions	Inlet (Subsonic)
		Outlet (Subsonic)
		Wall: No-Slip
		Wall: Adiabatic
	Timestep	Physical Time Scale
CFD-Post	Plots	Slice Plane
		Sphere Volume
	Other	Viewing the Mesh

4.2. Overview of the Problem to Solve

In this tutorial, you use a refined mesh to obtain a better solution to the Static Mixer problem created in *Simulating Flow in a Static Mixer Using CFX in Standalone Mode* (p. 7). You establish a general workflow for analyzing the flow of fluid into and out of a mixer. This tutorial uses a specific problem to teach the general approach taken when working with an existing mesh.

You start a new simulation in CFX-Pre and import the refined mesh. This tutorial introduces General Mode-the mode used for most tutorials-in CFX-Pre. The physics for this tutorial are the same as for *Simulating Flow in a Static Mixer Using CFX in Standalone Mode* (p. 7); therefore, you can import the physics settings used in that tutorial to save time.

4.3. Before You Begin

If this is the first tutorial you are working with, it is important to review the following topics before beginning:

- *Preparing a Working Directory and Starting ANSYS CFX in Standalone Mode* (p. 1)
- *Running ANSYS CFX Tutorials in ANSYS Workbench* (p. 2)
- *Changing the Display Colors* (p. 5)
- *Playing a Tutorial Session File* (p. 4)

4.4. Starting CFX-Pre

1. Prepare the working directory using the following files in the `examples` directory:
   - `StaticMixerRefMesh.gtm`
   - `StaticMixerRef.pre`
   - `StaticMixer.def`
   - `StaticMixer_001.res`
2. Set the working directory and start CFX-Pre.

   For details, see *Preparing a Working Directory and Starting ANSYS CFX in Standalone Mode* (p. 1)

4.5. Defining a Simulation using General Mode in CFX-Pre

After having completed meshing, CFX-Pre is used as a consistent and intuitive interface for the definition of complex CFD problems.

4.5.1. Workflow Overview

This section provides a brief summary of the topics so that you can see the workflow:

1. *Creating a New Case* (p. 65)
2. *Importing a Mesh* (p. 65)
3. *Importing Settings from Tutorial 1* (p. 65)
4. *Viewing Domain Settings* (p. 66)
5. *Viewing the Boundary Condition Setting* (p. 67)

6. *Defining Solver Parameters* (p. 67)
7. *Writing the CFX-Solver Input (.def) File* (p. 68)

As an alternative to these steps, you can set up the simulation automatically by following the procedures in *Playing the Session File and Starting CFX-Solver Manager* (p. 68).

To begin this tutorial and create a new simulation in CFX-Pre, continue from *Creating a New Case* (p. 65).

4.5.2. Creating a New Case

Before importing and working with a mesh, a simulation needs to be developed using General mode.

4.5.2.1. Procedure in Standalone Mode

1. In CFX-Pre, select **File** > **New Case**.
2. Select **General** in the **New Case** dialog box and click **OK**.
3. Select **File** > **Save Case As**.
4. Under **File name**, type `StaticMixerRef` and click **Save**.
5. Proceed to *Importing a Mesh* (p. 65).

4.5.3. Importing a Mesh

At least one mesh must be imported before physics are applied.

1. Select **File** > **Import** > **Mesh** or right-click `Mesh` and select **Import Mesh** > **Other**.
2. In the **Import Mesh** dialog box, select `StaticMixerRefMesh.gtm` from your working directory.

   This is a mesh that is more refined than the one used in Tutorial 1.
3. Click **Open**.

   The **Mesh** tree shows the regions in the `StaticMixerRefMesh.gtm` assembly[1] in a tree structure. The first tree branch displays the 3D regions and the level below each 3D region shows the 2D regions associated with it. The check box next to each item in the `Mesh` tree indicates the visibility status of the object in the viewer; you can click these to toggle visibility.
4. Right-click a blank area in the viewer and select **Predefined Camera** > **Isometric View (Z up)** from the shortcut menu.

4.5.4. Importing Settings from Tutorial 1

Because the physics and region names for this simulation are very similar to that for Tutorial 1, you can save time by importing the settings used there. We will be importing CCL from Tutorial 1 that contains settings that reference mesh regions. For example, the outlet boundary condition references the mesh region named `out`. In this tutorial, the name of the mesh regions are the same as in Tutorial 1, so you can import the CCL without error.

1. Select **File** > **Import** > **CCL**.

   The **Import CCL** dialog box appears.

[1] An *assembly* is a group of mesh regions that are topologically connected. Each assembly can contain only one mesh, but multiple assemblies are permitted.

2. Under **Import Method**, select **Replace**.

   Replace is useful if you have defined physics and want to update or replace them with newly imported physics.

3. Under **Files of type**, select `CFX-Solver input files (*def *res)`.
4. Select `StaticMixer.def` created in Tutorial 1. If you did not work through Tutorial 1, you can copy this file from the examples directory.
5. Click **Open**.
6. Select the **Outline** tab.

   The tree view displays a summary of the current simulation in a tree structure. Some items may be recognized from Tutorial 1; for example the boundary condition objects `in1`, `in2`, and `out`.

Note

- If you import CCL that references nonexistent mesh regions, you will get errors.
- A number of file types can be used as sources to import CCL, including:
  - Simulation files (`*.cfx`)
  - Results files (`*.res`)
  - CFX-Solver input files (`*.def`)
- The physics for a simulation can be saved to a CCL file at any time by selecting **File** > **Export** > **CCL**.

4.5.5. Viewing Domain Settings

It is useful to review the options available in General Mode.

Various domain settings can be set. These include:

- **Basic Settings**

  Specifies the location of the domain, coordinate frame settings and the fluids/solids that are present in the domain. You also reference pressure, buoyancy and whether the domain is stationary or rotating. Mesh motion can also be set.

- **Fluid Models**

  Sets models that apply to the fluid(s) in the domain, such as heat transfer, turbulence, combustion, and radiation models. An option absent in Tutorial 1 is **Turbulent Wall Functions**, which is set to **Scalable**. Wall functions model the flow in the near-wall region. For the k-epsilon turbulence model, you should always use scalable wall functions.

- **Initialization**

  Sets the initial conditions for the current domain only. This is generally used when multiple domains exist to allow setting different initial conditions in each domain, but can also be used to initialize single-domain simulations. Global initialization allows the specification of initial conditions for all domains that do not have domain-specific initialization.

1. On the **Outline** tree view, under `Simulation` > `Flow Analysis 1`, double-click `Default Domain`.

The domain `Default Domain` is opened for editing.

2. Click the **Basic Settings** tab and review, but do not change, the current settings.
3. Click **Fluid Models** and review, but do not change, the current settings.
4. Click **Initialization** and review, but do not change, the current settings.
5. Click **Close**.

4.5.6. Viewing the Boundary Condition Setting

For the k-epsilon turbulence model, you must specify the turbulent nature of the flow entering through the inlet boundary. For this simulation, the default setting of `Medium (Intensity = 5%)` is used. This is a sensible setting if you do not know the turbulence properties of the incoming flow.

1. Under `Default Domain`, double-click `in1`.
2. Click the **Boundary Details** tab and review the settings for **Flow Regime**, **Mass and Momentum**, **Turbulence** and **Heat Transfer**.
3. Click **Close**.

4.5.7. Defining Solver Parameters

Solver Control parameters control aspects of the numerical-solution generation process.

In Tutorial 1 you set some solver control parameters, such as **Advection Scheme** and **Timescale Control**, while other parameters were set automatically by CFX-Pre.

In this tutorial, **High Resolution** is used for the advection scheme. This is more accurate than the **Upwind Scheme** used in Tutorial 1. You usually require a smaller timestep when using this model. You can also expect the solution to take a higher number of iterations to converge when using this model.

1. Select **Insert** > **Solver** > **Solver Control** from the menu bar or click *Solver Control*  .
2. Apply the following **Basic Settings**

Setting	Value
Advection Scheme > Option	High Resolution
Convergence Control > Max. Iterations [1 (p. 67)]	150
Convergence Control > Fluid Timescale Control > Timescale Control	Physical Timescale
Convergence Control > Fluid Timescale Control > Physical Timescale	0.5 [s]

Footnote

1. If your solution does not meet the convergence criteria after this number of timesteps, the CFX-Solver will stop.

3. Click **Apply**.
4. Click the **Advanced Options** tab.

Tip

To select **Advanced Options** you may need to click the navigation icons next to the tabs to move 'forward' or 'backward' through the various tabs.

5. Ensure that **Global Dynamic Model Control** is selected.
6. Click **OK**.

4.5.8. Writing the CFX-Solver Input (.def) File

Once all boundaries are created you move from CFX-Pre into CFX-Solver.

The simulation file, `StaticMixerRef.cfx`, contains the simulation definition in a format that can be loaded by CFX-Pre, allowing you to complete (if applicable), restore, and modify the simulation definition. The simulation file differs from the CFX-Solver input file in two important ways:

- The simulation file can be saved at any time while defining the simulation.
- The CFX-Solver input file is an encapsulated set of meshes and CCL defining a solver run, and is a subset of the data in the simulation file.

1. Click *Define Run*  .

   The **Write Solver Input File** dialog box is displayed.
2. If required, set the path to your working directory.
3. Apply the following settings:

Setting	Value
File name	StaticMixerRef.def

4. Click **Save**.

   The CFX-Solver input file (`StaticMixerRef.def`) is created. CFX-Solver Manager automatically starts and, on the **Define Run** dialog box, the **Solver Input File** is set.
5. If you are notified in CFX-Pre that the file already exists, click **Overwrite**.
6. Quit CFX-Pre, saving the simulation (`.cfx`) file.
7. Proceed to *Obtaining a Solution Using CFX-Solver Manager* (p. 69).

4.5.9. Playing the Session File and Starting CFX-Solver Manager

If you have performed all the tasks in the previous steps, proceed directly to *Obtaining a Solution Using CFX-Solver Manager* (p. 69).

Two procedures are documented. Depending on your installation of CFX follow either the standalone procedure or the ANSYS Workbench procedure.

4.5.9.1. Procedure in Standalone

1. If required, launch CFX-Pre.

2. Select **Session** > **Play Tutorial**.
3. Select `StaticMixerRef.pre`.
4. Click **Open**.

   A CFX-Solver input file is written.
5. Select **File** > **Quit**.
6. Launch CFX-Solver Manager from CFX Launcher.
7. After CFX-Solver starts, select **File** > **Define Run**.
8. Under **CFX-Solver Input File**, click *Browse*  .
9. Select `StaticMixerRef.def`, located in the working directory.
10. Proceed to *Obtaining a Solution Using CFX-Solver Manager* (p. 69).

4.5.9.2. Procedure in ANSYS Workbench

1. If required, launch ANSYS Workbench.
2. Click **Empty Project**.
3. Select **File** > **Save** or click *Save*  .
4. Under **File name**, type `StaticMixerRef` and click **Save**.
5. Click **Start CFX-Pre**.
6. Select **Session** > **Play Tutorial**.
7. Select `StaticMixerRef.pre`.
8. Click **Open**.

   A CFX-Solver input file is written.
9. Click the **CFX-Solver** tab.
10. Select **File** > **Define Run**.
11. Under **CFX-Solver Input File**, click *Browse*  .
12. Select `StaticMixerRef.def`, located in the working directory.

4.6. Obtaining a Solution Using CFX-Solver Manager

Two windows are displayed when CFX-Solver Manager runs. There is an adjustable split between the windows which is oriented either horizontally or vertically, depending on the aspect ratio of the entire CFX-Solver Manager window (also adjustable).

4.6.1. Workflow Overview

This section provides a brief summary of the topics to follow as a general workflow:

1. *Starting the Run with an Initial Values File* (p. 70)
2. *Confirming Results* (p. 70)
3. *Moving from CFX-Solver to CFD-Post* (p. 71)

4.6.2. Starting the Run with an Initial Values File

In CFX-Solver Manager, the **Define Run** dialog box is visible and **Solver Input File** has automatically been set to the CFX-Solver input file from CFX-Pre: `StaticMixerRef.def`.

Configure the settings so that the results from Tutorial 1 (contained in `StaticMixer_001.res`) will be used to initialize the solution:

1. Select **Initial Values Specification**.
2. Set **Initial Values Specification** > **Initial Values** > **Initial Values 1** > **Initial Values 1 Settings** > **File Name** to `StaticMixer_001.res`.

   If you did not complete the first tutorial, you can use `StaticMixer_001.res` from your working directory.
3. Set **Use Mesh From** to `Solver Input File`.
4. Clear **Initial Values Specification** > **Continue History From**.

   This will cause the CFX-Solver to use the results in the `StaticMixer_001.res` file as a basic initial guess for the solution, and will cause the iteration count to start from 1 instead of from the last iteration number in the initial values file.
5. Click **Start Run**.

Note

Convergence information is plotted once the second outer loop iteration is complete.

4.6.3. Confirming Results

When the run is finished, specific information appears in the text window of CFX-Solver Manager.

To confirm that results interpolation was successful, look in the text window in CFX-Solver Manager. The following text appears before the convergence history begins:

```
+--------------------------------------------------------+
| Initial Conditions Supplied by Fields in the Input Files |
+--------------------------------------------------------+
```

This lists the variables that were interpolated from the results file.

After the final iteration, a message similar to the following content appears in a pop-up window:

```
StaticMixerRef_001 has completed normally.
Run concluded at: Fri Nov 27 11:57:57 2009
```

This indicates that CFX-Solver has successfully calculated the solution for the problem to the specified accuracy or has run out of coefficient loops.

1. In the **Solver Run Finished Normally** window, ensure that the check box next to **Post-Process Results** is cleared to prevent CFD-Post from launching at this time.
2. Click **OK**.
3. Review the CFX-Solver Manager's **Out File** tab for details on the run results.

4.6.4. Moving from CFX-Solver to CFD-Post

Once CFX-Solver has finished, you can use CFD-Post to review the finished results.

1. On the CFX-Solver Manager, select **Tools** > **Post-Process Results** or click *Post-Process Results*  in the toolbar.
2. In the **Start CFD-Post** dialog box, next to **Results File**, ensure that `StaticMixerRef_001.res` is set. If it is not, click *Browse*  and select `StaticMixerRef_001.res` (located in the working directory).
3. If using CFX-Solver in standalone mode, select **Shut down CFX-Solver Manager**.

   This forces Standalone CFX-Solver to close. This option is not required in Workbench.
4. Click **OK**.

   After a short pause, CFD-Post starts.

4.7. Viewing the Results in CFD-Post

In the following sections, you will explore the differences between the mesh and the results from this tutorial and Tutorial 1.

4.7.1. Creating a Slice Plane

More information exists for use by CFD-Post in this tutorial than in Tutorial 1 because the slice plane is more detailed.

Once a new slice plane is created it can be compared with Tutorial 1. There are three noticeable differences between the two slice planes.

- Around the edges of the mixer geometry there are several layers of narrow rectangles. This is the region where the mesh contains prismatic elements (which are created as inflation layers). The bulk of the geometry contains tetrahedral elements.
- There are more lines on the plane than there were in Tutorial 1. This is because the slice plane intersects with more mesh elements.
- The curves of the mixer are smoother than in Tutorial 1 because the finer mesh better represents the true geometry.

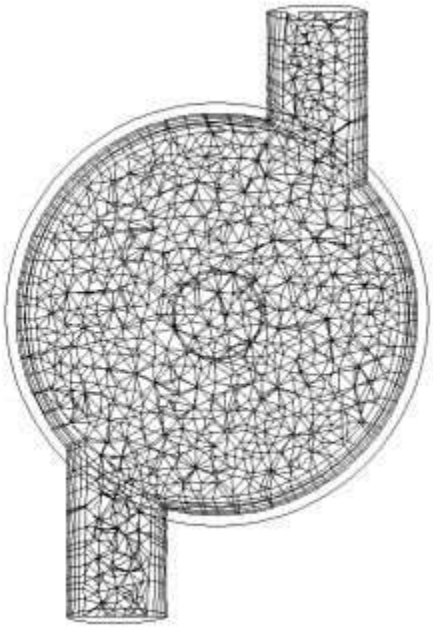
1. Right-click a blank area in the viewer and select **Predefined Camera** > **Isometric View (Z up)**.
2. From the menu bar, select **Insert** > **Location** > **Plane** or under **Location**, click **Plane**.
3. In the **Insert Plane** dialog box, type `Slice` and click **OK**.

   The **Geometry**, **Color**, **Render** and **View** tabs let you switch between settings.
4. Apply the following settings

Tab	Setting	Value
Geometry	Domains	Default Domain
	Definition > Method	XY Plane
	Definition > Z	1 [m]

Tab	Setting	Value
	Plane Type	Slice
Render	Show Faces	(Cleared)
	Show Mesh Lines	(Selected)

5. Click **Apply**.
6. Right-click a blank area in the viewer and select **Predefined Camera** > **View From +Z**.
7. If necessary, click *Zoom Box*  and zoom in on the geometry to view it in greater detail.
8. Compare the on-screen image with the equivalent picture from *Simulating Flow in a Static Mixer Using CFX in Standalone Mode* (p. 7) (in the section *Rendering Slice Planes* (p. 25)).



4.7.2. Coloring the Slice Plane

Here, you will color the plane by temperature.

1. Apply the following settings

Tab	Setting	Value
Color	Mode [1 (p. 73)]	Variable
	Variable	Temperature
	Range	Global
Render	Show Faces	(Selected)
	Show Mesh Lines	(Cleared)

Footnote

1. A mode setting of `Constant` would allow you to color the plane with a fixed color.

2. Click **Apply**.

4.7.3. Loading Results from Tutorial 1 for Comparison

In CFD-Post, you may load multiple results files into the same instance for comparison.

1. To load the results file from Tutorial 1, select **File** > **Load Results** or click *Load Results*  .
2. Be careful not to click **Open** until instructed to do so. In the **Load Results File** dialog box, select `StaticMixer_001.res` in the `<CFXROOT>\examples` directory or from your working directory if it has been copied.
3. On the right side of the dialog box, there are three frames:
   - **Case options**
   - **Additional actions**
   - **CFX run history and multi-configuration options**.

   Under **Case options**, select **Keep current cases loaded** and ensure that **Open in new view** is selected.
4. Under **Additional actions**, ensure that the **Clear user state before loading** check box is cleared.
5. Under **CFX run history and multi-configuration options**, ensure that **Load only the last results** is selected.
6. Click **Open** to load the results.

   In the tree view, there is now a second group of domains, meshes and boundary conditions with the heading `StaticMixer_001`.

   In the **3D Viewer**, there are two viewports named **View 1** and **View 2**; the former shows StaticMixer_001 and the latter shows StaticMixerRef_001.
7. Double-click the `Wireframe` object under `User Locations and Plots`.
8. In the **Definition** tab, set **Edge Angle** to `5 [degree]`.
9. Click **Apply**.
10. Click *Synchronize camera in displayed views*  so that all viewports maintain the same camera position.
11. Right-click a blank area in the viewer and select **Predefined Camera** > **Isometric View (Z up)**.

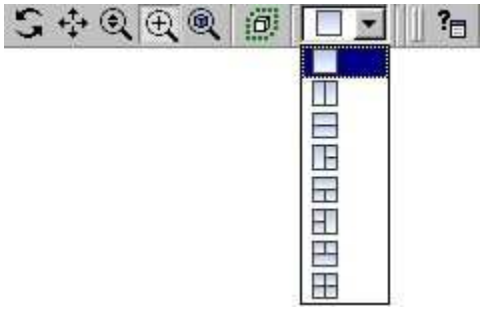
    Both meshes are now displayed in a line along the Y axis. Notice that one mesh is of a higher resolution than the other.
12. Set **Edge Angle** to `30 [degree]`.
13. Click **Apply**.

4.7.4. Comparing Slice Planes Using Multiple Views

The visibility status of each object is maintained separately for each viewport. This allows some planes to be shown while others are hidden. However, you can also use the *Synchronize visibility in displayed views*  to synchronize the visibility of objects that you add.

1. Under **User Locations and Plots**, select the check box beside `Slice`.
2. Right-click in the viewer and select **Predefined Camera** > **View From -Z**.

   Note the difference in temperature distribution.
3. In the viewer toolbar, click *Synchronize visibility in displayed views*  to deselect the option.
4. Click in **View 1** in the **3D Viewer**, then clear the visibility check box for `Slice` in the **Outline** tree view.
5. Click in **View 2**. Note that the visibility check box for `Slice` has been re-enabled as it describes the state of the plane for **View 2**. Clear the visibility check box for `Slice` in this view.
6. To return to a single viewport, select the option with a single rectangle.



7. Ensure that the visibility check box for `Slice` is cleared.
8. Right-click `StaticMixer_001` in the tree view and select **Unload**.

4.7.5. Viewing the Surface Mesh on the Outlet

In this part of the tutorial, you will view the mesh on the outlet. You will see five layers of inflated elements against the wall. You will also see the triangular faces of the tetrahedral elements closer to the center of the outlet.

1. Right-click a blank area in the viewer and select **Predefined Camera** > **Isometric View (Z up)**.
2. In the tree view, ensure that the visibility check box for `StaticMixerRef_001` > `Default Domain` > `out` is selected, then double-click `out` to open it for editing.

   Because the boundary location geometry was defined in CFX-Pre, the details view does not display a **Geometry** tab as it did for the planes.
3. Apply the following settings

Tab	Setting	Value
Render	Show Faces	(Cleared)
	Show Mesh Lines	(Selected)

Tab	Setting	Value
	Color Mode	User Specified
	Line Color	(Select any light color)

4. Click **Apply**.
5. Click *Zoom Box*  .
6. Zoom in on the geometry to view `out` in greater detail.
7. Click *Rotate*  on the **Viewing Tools** toolbar.
8. Rotate the image as required to clearly see the mesh.

4.7.6. Looking at the Inflated Elements in Three Dimensions

To show more clearly what effect inflation has on the shape of the elements, you will use volume objects to show two individual elements. The first element that will be shown is a normal tetrahedral element; the second is a prismatic element from an inflation layer of the mesh.

Leave the surface mesh on the outlet visible to help see how surface and volume meshes are related.

1. From the menu bar, select **Insert** > **Location** > **Volume** or, under **Location** click **Volume**.
2. In the **Insert Volume** dialog box, type `Tet Volume` and click **OK**.
3. Apply the following settings

Tab	Setting	Value
Geometry	Definition > Method	Sphere
	Definition > Point [1 (p. 75)]	0.08, 0, -2
	Definition > Radius	0.14 [m]
	Definition > Mode	Below Intersection
	Inclusive [2 (p. 75)]	(Cleared)
Color	Color	Red
Render	Show Faces > Transparency	0.3
	Show Mesh Lines	(Selected)
	Show Mesh Lines > Line Width	1
	Show Mesh Lines > Color Mode	User Specified
	Show Mesh Lines > Line Color	Grey

Footnotes

1. The z slider's minimum value corresponds to the minimum z value of the entire geometry, which, in this case, occurs at the outlet.
2. Only elements that are entirely contained within the sphere volume will be included.

4. Click **Apply** to create the volume object.
5. Right-click `Tet Volume` and choose **Duplicate**.
6. In the **Duplicate Tet Volume** dialog box, type `Prism Volume` and click **OK**.
7. Double-click `Prism Volume`.
8. Apply the following settings

Tab	Setting	Value
Geometry	Definition > Point	-0.22, 0.4, -1.85
	Definition > Radius	0.206 [m]
Color	Color	Orange

9. Click **Apply**.

4.7.7. Viewing the Surface Mesh on the Mixer Body

1. Double-click the `Default Domain Default` object.
2. Apply the following settings

Tab	Setting	Value
Render	Show Faces	(Selected)
	Show Mesh Lines	(Selected)
	Line Width	2

3. Click **Apply**.

4.7.8. Viewing the Layers of Inflated Elements on a Plane

You will see the layers of inflated elements on the wall of the main body of the mixer. Within the body of the mixer, there will be many lines that are drawn wherever the face of a mesh element intersects the slice plane.

1. From the menu bar, select **Insert** > **Location** > **Plane** or under **Location**, click **Plane**.
2. In the **Insert Plane** dialog box, type `Slice 2` and click **OK**.
3. Apply the following settings

Tab	Setting	Value
Geometry	Definition > Method	YZ Plane
	Definition > X	0 [m]
Render	Show Faces	(Cleared)
	Show Mesh Lines	(Selected)

4. Click **Apply**.
5. Turn off the visibility of all objects except `Slice 2`.
6. To see the plane clearly, right-click in the viewer and select **Predefined Camera** > **View From -X**.

4.7.9. Viewing the Mesh Statistics

You can use the Report Viewer to check the quality of your mesh. For example, you can load a `.def` file into CFD-Post and check the mesh quality before running the `.def` file in the solver.

1. Click the **Report Viewer** tab (located below the viewer window).

   A report appears. Look at the table shown in the "Mesh Report" section.
2. Double-click `Report` > `Mesh Report` in the **Outline** tree view.
3. In the **Mesh Report** details view, select **Statistics** > **Maximum Face Angle**.
4. Click **Refresh Preview**.

   Note that a new table, showing the maximum face angle for all elements in the mesh, has been added to the "Mesh Report" section of the report. The maximum face angle is reported as 148.95°.

As a result of generating this mesh statistic for the report, a new variable, `Maximum Face Angle`, has been created and stored at every node. This variable will be used in the next section.

4.7.10. Viewing the Mesh Elements with Largest Face Angle

In this section, you will visualize the mesh elements that have a `Maximum Face Angle` value greater than 140°.

1. Click the **3D Viewer** tab (located below the viewer window).
2. Right-click a blank area in the viewer and select **Predefined Camera** > **Isometric View (Z up)**.
3. In the **Outline** tree view, select the visibility check box of `Wireframe`.
4. From the menu bar, select **Insert** > **Location** > **Volume** or under **Location**, click **Volume**.
5. In the **Insert Volume** dialog box, type `Max Face Angle Volume` and click **OK**.
6. Apply the following settings

Tab	Setting	Value
Geometry	Definition > Method	Isovolume
	Definition > Variable	Maximum Face Angle [1 (p. 77)]
	Definition > Mode	Above Value
	Definition > Value	140 [degree]
	Inclusive [2 (p. 77)]	(Selected)

Footnotes

1. Select `Maximum Face Angle` from the larger list of variables available by clicking ... to the right of the **Variable** box.
2. This includes any elements that have at least one node with a variable value greater than or equal to the given value.

7. Click **Apply**.

   The volume object appears in the viewer.

4.7.11. Viewing the Mesh Elements with Largest Face Angle Using a Point

Next, you will create a point object to show a node that has the maximum value of `Maximum Face Angle`. The point object will be represented by a 3D yellow crosshair symbol. In order to avoid obscuring the point object with the volume object, you may want to turn off the visibility of the latter.

1. From the menu bar, select **Insert** > **Location** > **Point** or under **Location**, click **Point**.
2. Click **OK** to use the default name.
3. Apply the following settings

Tab	Setting	Value
Geometry	Definition > Method	Variable Maximum
	Definition > Location	Default Domain
	Definition > Variable	Maximum Face Angle
Symbol	Symbol Size	2

4. Click **Apply**.

4.7.12. Quitting CFD-Post

1. When you are finished, select **File** > **Quit** to exit CFD-Post.
2. If prompted by a dialog box, save the state at your discretion.

Chapter 5: Flow in a Process Injection Mixing Pipe

This tutorial includes:

5.1. Tutorial Features

In this tutorial you will learn about:

- Applying a profile boundary using data stored in a file.
- Visualizing the velocity on a boundary in CFX-Pre.
- Using the CFX Expression Language (CEL) to describe temperature dependent fluid properties in CFX-Pre.
- Using the k-epsilon turbulence model.
- Using streamlines in CFD-Post to track flow through the domain.

Component	Feature	Details
CFX-Pre	User Mode	General Mode
	Analysis Type	Steady State
	Fluid Type	General Fluid
	Domain Type	Single Domain
	Turbulence Model	k-Epsilon
	Heat Transfer	Thermal Energy
	Boundary Conditions	Boundary Profile Visualization
		Inlet (Profile)
		Inlet (Subsonic)
		Outlet (Subsonic)
		Wall: No-Slip
		Wall: Adiabatic
	CEL (CFX Expression Language)	
	Timestep	Physical Time Scale
CFD-Post	Plots	Default Locators
		Outline Plot (Wireframe)

Component	Feature	Details
		Slice Plane
		Streamline
	Other	Changing the Color Range
		Expression Details View
		Legend
		Viewing the Mesh

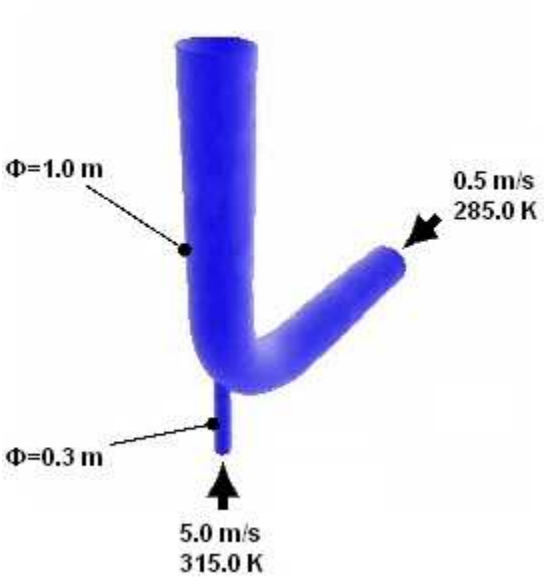
5.2. Overview of the Problem to Solve

The goal of this tutorial is to understand the general approach taken when working with an existing mesh. In this tutorial, you will go through the three main steps when solving a problem, which are defining a simulation using General Mode in CFX-Pre, obtaining a solution using CFX-Solver Manager and viewing the results in CFD-Post.

The injection mixing pipe, common in the process industry, is composed of two pipes: one with a larger diameter than the other. Analyzing and optimizing the mixing process is often critical for many chemical processes. CFD is useful not only in identifying problem areas (where mixing is poor), but also in testing new designs before they are implemented.

The geometry for this example consists of a circular pipe of diameter 1.0 m with a 90° bend, and a smaller pipe of diameter 0.3 m which joins with the main pipe at an oblique angle. Water at 315.0 K enters in the 0.3 m diameter pipe at a rate of 5.0 m/s while water at 285.0 K enters in the 1.0 m diameter pipe at a rate of 0.5 m/s.

Figure 5.1 Injection Mixing Pipe



In this tutorial, you will establish a general workflow for analyzing the flow of the water fluid into and out of an injection pipe. First, a simulation will be created and an existing mesh will be imported in CFX-Pre. A viscosity expression will also be created, and will be used to modify the water properties later on in this tutorial to increase the solution accuracy. Finally, initial values will be set and a solution will be found using CFX-Solver Manager. The results will then be viewed in CFD-Post. Streamlines originating from the main inlet will be generated to show the flow of the water into and out of the injection pipe.

5.3. Before You Begin

If this is the first tutorial you are working with, it is important to review the following topics before beginning:

- *Preparing a Working Directory and Starting ANSYS CFX in Standalone Mode* (p. 1)
- *Running ANSYS CFX Tutorials in ANSYS Workbench* (p. 2)
- *Changing the Display Colors* (p. 5)
- *Playing a Tutorial Session File* (p. 4)

5.4. Starting CFX-Pre

1. Prepare the working directory using the following files in the `examples` directory:
   - `InjectMixer.pre`
   - `InjectMixer_velocity_profile.csv`
   - `InjectMixerMesh.gtm`
2. Set the working directory and start CFX-Pre.

   For details, see *Preparing a Working Directory and Starting ANSYS CFX in Standalone Mode* (p. 1)

5.5. Defining a Simulation using General Mode in CFX-Pre

If you want to set up the simulation automatically using a tutorial session file, run `InjectMixer.pre`. For details, see *Playing a Tutorial Session File* (p. 4). Then proceed to *Obtaining a Solution Using CFX-Solver Manager* (p. 88).

If you want to set up the simulation manually, continue with the following steps.

5.5.1. Workflow Overview

This section provides a brief summary of the topics to follow as a general workflow:

1. *Creating a New Case* (p. 82)
2. *Importing a Mesh* (p. 82)
3. *Setting Temperature-Dependent Material Properties* (p. 82)
4. *Plotting an Expression* (p. 83)
5. *Evaluating an Expression* (p. 84)
6. *Modify Material Properties* (p. 84)
7. *Creating the Domain* (p. 84)
8. *Creating the Side Inlet Boundary* (p. 85)
9. *Creating the Main Inlet Boundary* (p. 86)
10. *Creating the Main Outlet Boundary* (p. 87)
11. *Setting Initial Values* (p. 87)
12. *Setting Solver Control* (p. 87)
13. *Writing the CFX-Solver Input (.def) File* (p. 88)

5.5.2. Creating a New Case

Before importing and working with a mesh, a simulation needs to be started using General Mode.

5.5.2.1. Procedure in Standalone

1. In CFX-Pre, select **File** > **New Case**.
2. Ensure **General** is selected and click **OK**.
3. Select **File** > **Save Case As**.
4. Under **File name**, type `InjectMixer`.
5. Click **Save**.
6. Proceed to *Importing a Mesh* (p. 82).

5.5.3. Importing a Mesh

The following steps will demonstrate how to import a mesh.

1. Edit `Case Options` > `General` in the **Outline** tree view and ensure that **Automatic Default Domain** is turned off.

   Default Domain generation should be turned off because you will create a new domain manually, later in this tutorial.
2. Select **File** > **Import** > **Mesh**.
3. From your working directory, select `InjectMixerMesh.gtm`.
4. Click **Open**.
5. Right-click a blank area in the viewer and select **Predefined Camera** > **Isometric View (Y up)** from the shortcut menu.

5.5.4. Setting Temperature-Dependent Material Properties

Viscosity varies with temperature, which implies that the water will behave differently when coming through the 1.0 m and the 0.3 m diameter pipes. In the following steps, you will create an expression for viscosity as a function of temperature. This expression will then be used to modify the properties of the library material: `Water` to increase the accuracy of the solution. By setting temperature-dependent material properties, Viscosity will be made to vary linearly with temperature between the following conditions:

- μ=1.8E-03 N s m^{-2} at T=275.0 K
- μ=5.45E-04 N s m^{-2} at T=325.0 K

The variable T (Temperature) is a CFX System Variable recognized by CFX-Pre, denoting static temperature. All variables, expressions, locators, functions, and constants can be viewed by double-clicking the appropriate entry (such as `Additional Variables` or `Expressions`) in the tree view.

All expressions must have consistent units. You should be careful if using temperature in an expression with units other than `[K]`.

The **Expressions** tab lets you define, modify, evaluate, plot, copy, delete and browse through expressions used within CFX-Pre.

1. From the main menu, select **Insert** > **Expressions, Functions and Variables** > **Expression**.
2. In the **New Expression** dialog box, type `Tupper`.
3. Click **OK**.

   The details view for the `Tupper` equation is displayed.
4. Under **Definition**, type `325 [K]`.
5. Click **Apply** to create the expression.

   The expression is added to the list of existing expressions.
6. Right-click in the **Expressions** workspace and select **Insert** > **Expression**.
7. In the **New Expression** dialog box, type `Tlower`.
8. Click **OK**.
9. Under **Definition**, type `275 [K]`.
10. Click **Apply** to create the expression.

    The expression is added to the list of existing expressions.
11. Create expressions for `Visupper`, `Vislower` and `VisT` using the following values.

Name	Definition
Visupper	5.45E-04 [N s m^-2]
Vislower	1.8E-03 [N s m^-2]
VisT	Vislower+(Visupper-Vislower)*(T-Tlower)/(Tupper-Tlower)

5.5.5. Plotting an Expression

1. Right-click `VisT` in the **Expressions** tree view, and then select **Edit**.

   The **Expressions** details view for `VisT` appears.

   Tip

   Alternatively, double-clicking the expression also opens the **Expressions** details view.

2. Click the **Plot** tab and apply the following settings

Tab	Setting	Value
Plot	Number of Points	10
	T	(Selected)
	Start of Range	275 [K]
	End of Range	325 [K]

3. Click **Plot Expression**.

   A plot showing the variation of the expression `VisT` with the variable `T` is displayed.

5.5.6. Evaluating an Expression

1. Click the **Evaluate** tab.
2. In T, type `300 [K]`.

   This is between the start and end range defined in the last module.
3. Click **Evaluate Expression**.

   A value of around 0.0011[kg m^-1 s^-1] for `VisT` at the given value of T appears in the **Value** field.

5.5.7. Modify Material Properties

As mentioned earlier in this tutorial, the default material properties of `Water` will be modified using the Viscosity expression to increase the accuracy of the solution.

1. Click the **Outline** tab.
2. Double-click `Water` under `Materials` to display the **Basic Settings** tab.
3. Click the **Material Properties** tab.
4. Expand **Transport Properties**.
5. Select **Dynamic Viscosity**.
6. Under **Dynamic Viscosity**, click in **Dynamic Viscosity**.
7. Click *Enter Expression*  .
8. Enter the expression `VisT` into the data box.
9. Click **OK**.

5.5.8. Creating the Domain

The domain will be set to use the thermal energy heat transfer model, and the $k - \varepsilon$ (k-epsilon) turbulence model.

Both **Basic Settings** and **Fluid Models** are changed in this module. The **Initialization** tab is for setting domain-specific initial conditions, which are not used in this tutorial. Instead, global initialization is used to set the starting conditions.

1. Ensure that no default domain is present under `Flow Analysis 1`. If a default domain is present, right-click it and select **Delete**.
2. Select **Insert** > **Domain** from the main menu or click *Domain*  .
3. In the **Insert Domain** dialog box, type `InjectMixer`.
4. Click **OK**.
5. Apply the following settings

Tab	Setting	Value
Basic Settings	Location and Type > Location	B1.P3

Tab	Setting	Value
	Fluid and Particle Definitions	Fluid 1
	Fluid and Particle Definitions > Fluid 1 > Material	Water
	Domain Models > Pressure > Reference Pressure	0 [atm]

6. Click **Fluid Models**.
7. Apply the following settings

Setting	Value
Heat Transfer > Option	Thermal Energy

8. Click **OK**.

5.5.9. Creating the Side Inlet Boundary

The side inlet boundary needs to be defined.

1. Select **Insert** > **Boundary** from the main menu or click *Boundary*  .
2. Set **Name** to `side inlet`.

Note

A boundary named after a region will use that region as its location by default.

---

3. Click **OK**.
4. Apply the following settings

Tab	Setting	Value
Basic Settings	Boundary Type	Inlet
	Location	side inlet
Boundary Details	Mass and Momentum > Option	Normal Speed
	Mass and Momentum > Normal Speed	5 [m s^-1]
	Heat Transfer > Option	Static Temperature
	Heat Transfer > Static Temperature	315 [K]

5. Click **OK**.

5.5.10. Creating the Main Inlet Boundary

The main inlet boundary for the large pipe needs to be defined. This inlet is defined using a velocity profile found in the examples directory. Profile data needs to be initialized before the boundary can be created.

You will create a plot showing the velocity profile data, marked by higher velocities near the center of the inlet, and lower velocities near the inlet walls.

1. Select **Tools** > **Initialize Profile Data**.
2. Under **Data File**, click *Browse*  .
3. From your working directory, select `InjectMixer_velocity_profile.csv`.
4. Click **Open**.
5. Click **OK**.

   The profile data is read into memory.
6. Select **Insert** > **Boundary** from the main menu or click *Boundary*  .
7. Set name **Name** to `main inlet`.
8. Click **OK**.
9. Apply the following settings

Tab	Setting	Value
Basic Settings	Boundary Type	Inlet
	Location	main inlet
	Profile Boundary Conditions > Use Profile Data	(Selected)
	Profile Boundary Setup > Profile Name	main inlet

10. Click **Generate Values**.

    This causes the profile values of U, V, W to be applied at the nodes on the main inlet boundary, and U, V, W entries to be made in **Boundary Details**. To later modify the velocity values at the main inlet and reset values to those read from the BC Profile file, revisit **Basic Settings** for this boundary and click **Generate Values**.
11. Apply the following settings

Tab	Setting	Value
Boundary Details	Flow Regime > Option	Subsonic
	Turbulence > Option	Medium (Intensity = 5%)
	Heat Transfer > Option	Static Temperature
	Heat Transfer > Static Temperature	285 [K]
Plot Options	Boundary Contour	(Selected)

Tab	Setting	Value
	Boundary Contour > Profile Variable	W

12. Click **OK**.
13. Zoom into the main inlet to view the inlet velocity contour.

5.5.11. Creating the Main Outlet Boundary

In this module you create the outlet boundary. All other surfaces which have not been explicitly assigned a boundary will remain in the `InjectMixer Default` object, which is shown in the tree view. This boundary uses a `No-Slip Adiabatic Wall` by default.

1. Select **Insert** > **Boundary** from the main menu or click *Boundary*  .
2. Set **Name** to `outlet`.
3. Click **OK**.
4. Apply the following settings

Tab	Setting	Value
Basic Settings	Boundary Type	Outlet
	Location	outlet
Boundary Details	Flow Regime > Option	Subsonic
	Mass and Momentum > Option	Average Static Pressure
	Relative Pressure	0 [Pa]

5. Click **OK**.

5.5.12. Setting Initial Values

For this tutorial, the initial values will be set automatically. An automatic guess is sufficient for this particular problem.

1. Click *Global Initialization*  and review, but do not change, the current settings.
2. Click **Close**.

5.5.13. Setting Solver Control

1. Click *Solver Control*  .
2. Apply the following settings

Tab	Setting	Value
Basic Settings	Advection Scheme > Option	Specified Blend Factor

Tab	Setting	Value
	Advection Scheme > Blend Factor	0.75
	Convergence Control > Max. Iterations	50
	Convergence Control > Fluid Timescale Control > Timescale Control	Physical Timescale
	Convergence Control > Fluid Timescale Control > Physical Timescale	2 [s][a]
	Convergence Criteria > Residual Type	RMS
	Convergence Criteria > Residual Target	1.E-4[b]

[a]The physical timescale that will be setup is derived from the pipe diameter (1 m) and the rate at which the water flows in the pipe (0.5 m/s).
[b]An RMS value of at least 1.E-5 is usually required for adequate convergence, but the default value is sufficient for demonstration purposes.

3. Click **OK**.

5.5.14. Writing the CFX-Solver Input (.def) File

Once the problem has been defined you move from General Mode into CFX-Solver.

1. Click *Define Run*  .
2. Apply the following settings:

Setting	Value
File name	InjectMixer.def

3. Click **Save**.

   The CFX-Solver input file (`InjectMixer.def`) is created. CFX-Solver Manager automatically starts and, on the **Define Run** dialog box, the **Solver Input File** is set.
4. When you are finished, select **File** > **Quit** in CFX-Pre.
5. Click **Save & Quit** if prompted, to save `InjectMixer.cfx`
6. Proceed to *Obtaining a Solution Using CFX-Solver Manager* (p. 88).

5.6. Obtaining a Solution Using CFX-Solver Manager

At this point, CFX-Solver Manager is running, and the **Define Run** dialog box is displayed, with the CFX-Solver input file set.

1. Click **Start Run**.

2. When the run ends, ensure that the check box next to **Post-Process Results** is cleared and click **OK** to close the dialog box.

5.6.1. Moving from CFX-Solver Manager to CFD-Post

1. Select **Tools** > **Post-Process Results** or click *Post-Process Results*  .
2. If using CFX-Solver Manager in standalone mode, optionally select **Shut down CFX-Solver Manager**.
3. Click **OK**.

5.7. Viewing the Results in CFD-Post

When CFD-Post starts, the viewer and **Outline** workspace display by default.

5.7.1. Workflow Overview

This section provides a brief summary of the topics to follow as a general workflow:

1. *Modifying the Outline of the Geometry* (p. 89)
2. *Creating and Modifying Streamlines Originating from the Main Inlet* (p. 89)
3. *Modifying Streamline Color Ranges* (p. 90)
4. *Coloring Streamlines with a Constant Color* (p. 90)
5. *Creating Streamlines Originating from the Side Inlet* (p. 91)
6. *Examining Turbulence Kinetic Energy* (p. 91)

5.7.2. Modifying the Outline of the Geometry

Throughout this and the following examples, use your mouse and the **Viewing Tools** toolbar to manipulate the geometry as required at any time.

1. In the tree view, double-click `Wireframe`.
2. Set the **Edge Angle** to `15 [degree]`.
3. Click **Apply**.

5.7.3. Creating and Modifying Streamlines Originating from the Main Inlet

When you complete this module you will see streamlines (mainly blue and green) starting at the main inlet of the geometry and proceeding to the outlet. Above where the side pipe meets the main pipe, there is an area where the flow re-circulates rather than flowing roughly tangent to the direction of the pipe walls.

1. Select **Insert** > **Streamline** from the main menu or click *Streamline*  .
2. Under **Name**, type `MainStream`.
3. Click **OK**.
4. Apply the following settings

Tab	Setting	Value
Geometry	Type	3D Streamline

Tab	Setting	Value
	Definition > Start From	main inlet

5. Click **Apply**.
6. Right-click a blank area in the viewer, select **Predefined Camera** from the shortcut menu, then select **Isometric View (Y up)**.

   The pipe is displayed with the main inlet in the bottom right of the viewer.

5.7.4. Modifying Streamline Color Ranges

You can change the appearance of the streamlines using the **Range** setting on the **Color** tab.

1. Under **User Locations and Plots**, modify the streamline object **MainStream** by applying the following settings

Tab	Setting	Value
Color	Range	Local

2. Click **Apply**.

   The color map is fitted to the range of velocities found along the streamlines. The streamlines therefore collectively contain every color in the color map.
3. Apply the following settings

Tab	Setting	Value
Color	Range	User Specified
	Min	0.2 [m s^-1]
	Max	2.2 [m s^-1]

Note

Portions of streamlines that have values outside the range shown in the legend are colored according to the color at the nearest end of the legend. When using tubes or symbols (which contain faces), more accurate colors are obtained with lighting turned off.

4. Click **Apply**.

   The streamlines are colored using the specified range of velocity values.

5.7.5. Coloring Streamlines with a Constant Color

1. Apply the following settings

Tab	Setting	Value
Color	Mode	Constant
	Color	(Green)

Color can be set to green by selecting it from the color pallet, or by repeatedly clicking on the color box until it cycles through to the default green color.

2. Click **Apply**.

5.7.6. Creating Streamlines Originating from the Side Inlet

The following steps illustrate using this feature to add a streamline object that originates at the side inlet.

1. Right-click `MainStream` and select **Duplicate** from the shortcut menu.
2. In the **Name** window, type `SideStream`.
3. Click **OK**.
4. Double-click the newly created streamline, `SideStream`.
5. Apply the following settings

Tab	Setting	Value
Geometry	Definition > Start From	side inlet
Color	Mode	Constant
	Color	(Red)

6. Click **Apply**.

   Red streamlines appear, starting from the side inlet.

7. For better view, select **Isometric View (Y up)**.

5.7.7. Examining Turbulence Kinetic Energy

Away from walls, turbulence kinetic energy has an influence on the level of mixing. A plane will be created to view the Turbulence Kinetic Energy variable within the domain.

Note

This module has multiple changes compiled into single steps in preparation for other tutorials that provide fewer specific instructions.

1. Turn off the visibility of both the `MainStream` and the `SideStream` objects.
2. Create a plane named `Plane 1` that is normal to X and passing through the X = 0 Point. To do so, specific instructions follow.
   1. From the main menu, select **Insert** > **Location** > **Plane** and click **OK**.
   2. In the **Details** view set **Definition > Method** to `YZ Plane` and **X** to `0 [m]`.
   3. Click **Apply**.
3. Color the plane using the variable `Turbulence Kinetic Energy`, to show regions of high turbulence. To do so, apply the settings below.

Tab	Setting	Value
Color	Mode	Variable

Tab	Setting	Value
	Variable	Turbulence Kinetic Energy

4. Click **Apply**.
5. Experiment with other variables to color this plane (for example, `Temperature` to show the temperature mixing of the two streams).

   Commonly used variables are in the drop-down menu. A full list of available variables can be viewed by clicking  next to the **Variable** data box.

5.7.8. Quitting CFD-Post

1. When you are finished, select **File** > **Quit** to exit CFD-Post.
2. If prompted by a dialog box, save the state at your discretion.

Chapter 6: Flow from a Circular Vent

This tutorial includes:

6.1. Tutorial Features

In this tutorial you will learn about:

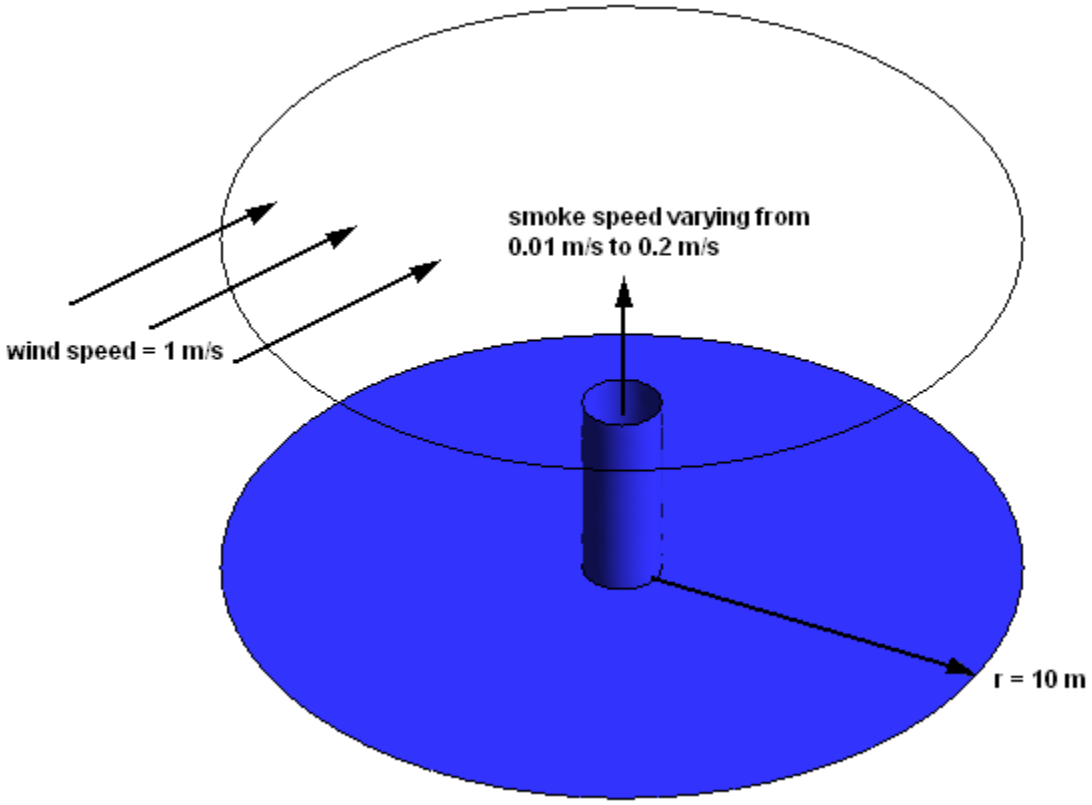
- Setting up a transient problem in CFX-Pre.
- Using an opening type boundary in CFX-Pre.
- Making use of multiple configurations in CFX-Pre
- Modeling smoke using additional variables in CFX-Pre.
- Visualizing a smoke plume using an Isosurface in CFD-Post.
- Creating an image for printing, and generating a movie in CFD-Post.

Component	Feature	Details
CFX-Pre	User Mode	General Mode
	Analysis Type	Steady State
		Transient
	Fluid Type	General Fluid
	Domain Type	Single Domain
	Turbulence Model	k-Epsilon
	Boundary Conditions	Inlet (Subsonic)
		Opening
		Wall: No-Slip
	Timestep	Auto Time Scale
		Transient Example
	Configuration	Multiple
	Transient Results File	
CFD-Post	Plots	Animation
		Isosurface
	Other	Auto Annotation

Component	Feature	Details
		Movie Generation
		Printing
		Time Step Selection
		Title/Text
		Transient Animation

6.2. Overview of the Problem to Solve

In this example, a chimney stack releases smoke that is dispersed into the atmosphere with an oncoming side wind of 1 m/s. The turbulence will be set to intensity and length scale with a value of 0.05, which corresponds to 5% turbulence, a medium level intensity, and with an eddy length scale value of 0.25 m. The goal of this tutorial is to model the dispersion of the smoke from the chimney stack over time. Unlike previous tutorials, which were steady-state, this example is time-dependent. Initially, no smoke is being released. Subsequently, the chimney starts to release smoke. As a postprocessing exercise, you produce an animation that illustrates how the plume of smoke develops with time.



6.3. Before You Begin

If this is the first tutorial you are working with, it is important to review the following topics before beginning:

- *Preparing a Working Directory and Starting ANSYS CFX in Standalone Mode* (p. 1)
- *Running ANSYS CFX Tutorials in ANSYS Workbench* (p. 2)
- *Changing the Display Colors* (p. 5)
- *Playing a Tutorial Session File* (p. 4)

6.4. Starting CFX-Pre

1. Prepare the working directory using the following files in the `examples` directory:
   - `CircVent.pre`
   - `CircVentMesh.gtm`
2. Set the working directory and start CFX-Pre.

   For details, see *Preparing a Working Directory and Starting ANSYS CFX in Standalone Mode* (p. 1)

6.5. Defining a Multiple-Analysis Simulation in CFX-Pre

If you want to set up the simulation automatically using a tutorial session file, run `CircVent.pre`. For details, see *Playing a Tutorial Session File* (p. 4). Then proceed to *Obtaining Solutions to the Steady-State and Transient Configurations* (p. 106).

If you want to set up the simulation manually, proceed to the following steps:

This section describes the step-by-step definition of the flow physics in CFX-Pre for a simulation with two analyses. First is a steady-state analysis with no smoke being produced by the chimney. The second analysis takes the setup for the steady state and adapts it for a transient analysis. The results from the steady-state analysis will be used as the initial guess for the transient analysis.

1. In CFX-Pre, select **File** > **New Case**.
2. Select **General** and click **OK**.
3. Select **File** > **Save Case As**.
4. Set **File name** to `CircVent`.
5. Click **Save**.

6.5.1. Importing the Mesh

1. Edit `Case Options` > `General` in the **Outline** tree view, clear **Automatic Default Domain**, and click **OK**.

   Default Domain generation is turned off so that you can create a new domain manually later in this tutorial.
2. Select **File** > **Import** > **Mesh**.
3. From your working directory, select `CircVentMesh.gtm`.
4. Click **Open**.
5. Right-click a blank area in the viewer and select **Predefined Camera** > **Isometric View (Z up)** from the shortcut menu.

6.5.2. Creating an Additional Variable

In this tutorial an Additional Variable (non-reacting scalar component) will be used to model the dispersion of smoke from the vent.

Note

While smoke is not required for the steady-state simulation, including it here prevents you from having to set up time value interpolation in the transient analysis.

1. From the menu bar, select **Insert** > **Expressions, Functions and Variables** > **Additional Variable**.
2. Set **Name** to `smoke`.
3. Click **OK**.
4. Set **Variable Type** to `Volumetric`.
5. Set **Units** to `[kg m^-3]`.
6. Click **OK**.

6.5.3. Renaming the Analysis

To rename the existing analysis:

1. From the **Outline** Tree, right-click `Simulation` > `Flow Analysis 1` and click **Rename**.
2. Rename the analysis to `Steady State Analysis`.

6.5.4. Creating the Domain

You will create a fluid domain that includes support for smoke as an Additional Variable.

6.5.4.1. To Create a New Domain

1. Select **Insert** > **Domain** from the menu bar, or click *Domain*  , then set the name to `CircVent` and click **OK**.
2. Apply the following settings

Tab	Setting	Value
Basic Settings	Location and Type > Location	B1.P3
	Fluid and Particle Definitions	Fluid 1
	Fluid and Particle Definitions > Fluid 1 > Material	Air at 25 C
	Domain Models > Pressure > Reference Pressure	0 [atm]
Fluid Models	Heat Transfer > Option	None
	Additional Variable Models > Additional Variable	smoke
	Additional Variable Models > Additional Variable > smoke	(Selected)
	Additional Variable Models > Additional Variable > smoke > Kinematic Diffusivity	(Selected)

Tab	Setting	Value
	Additional Variable Models > Additional Variable > smoke > Kinematic Diffusivity > Kinematic Diffusivity	1.0E-5 [m^2 s^-1] [1 (p. 97)]
1. 1.0E-5 [m^2 s^-1] is a representative kinematic diffusivity value for smoke in air.		

3. Click **OK**.

6.5.5. Creating the Boundaries

This is an example of external flow, since fluid is flowing *over* an object and not *through* an enclosure such as a pipe network (which would be an example of internal flow). In external flow problems, some inlets will be made sufficiently large that they do not affect the CFD solution. However, the length scale values produced by the **Default Intensity and AutoCompute Length Scale** option for turbulence are based on inlet size. They are appropriate for internal flow problems and particularly, cylindrical pipes. In general, you need to set the turbulence intensity and length scale explicitly for large inlets in external flow problems. If you do not have a value for the length scale, you can use a length scale based on a typical length of the object over which the fluid is flowing. In this case, you will choose a turbulence length scale which is one-tenth of the diameter of the vent.

For parts of the boundary where the flow direction changes, or is unknown, an opening boundary can be used. An opening boundary allows flow to both enter and leave the fluid domain during the course of the analysis.

6.5.5.1. Inlet Boundary

You will create the inlet boundary with velocity components set consistently with the problem description.

1. Select **Insert** > **Boundary** from the menu bar or click *Boundary*  .
2. Set **Name** to `Wind`.
3. Click **OK**.
4. Apply the following settings

Tab	Setting	Value
Basic Settings	Boundary Type	Inlet
	Location	Wind
Boundary Details	Mass And Momentum > Option	Cart. Vel. Components
	Mass and Momentum > U	1 [m s^-1]
	Mass and Momentum > V	0 [m s^-1]
	Mass and Momentum > W	0 [m s^-1]
	Turbulence > Option	Intensity and Length Scale
	Turbulence > Fractional Intensity	0.05 [1 (p. 98)]
	Turbulence > Eddy Length Scale	0.25 [m] [1 (p. 98)]

Tab	Setting	Value
	Additional Variables > smoke > Option	Value
	Additional Variables > smoke > Add. Var. Value	0 [kg m^-3] [2 (p. 98)]
1. From the problem description. 2. The smoke value that will be set up corresponds to no smoke at the inlet.		

5. Click **OK**.

Note

The boundary marker vectors used to display boundary conditions (inlets, outlets, openings) are normal to the boundary surface regardless of the actual direction specification. To plot vectors in the direction of flow, select **Boundary Vector** under the **Plot Options** tab for the inlet boundary, and on the **Labels and Markers Options** tab (accessible from **Case Options** > **Labels and Markers** on the **Outline** tree view), ensure that **Settings** > **Show Boundary Markers** is selected and **Show Inlet Markers** is cleared.

6.5.5.2. Opening Boundary

You will create an opening boundary with pressure and flow direction specified. If fluid enters the domain through the opening, it should have turbulence intensity and length scale, as well as smoke concentration, set to the same values as for the inlet.

1. Select **Insert** > **Boundary** from the menu bar or click *Boundary*  .
2. Set **Name** to `Atmosphere`.
3. Click **OK**.
4. Apply the following settings

Tab	Setting	Value
Basic Settings	Boundary Type	Opening
	Location	Atmosphere
Boundary Details	Mass And Momentum > Option	Opening Pres. and Dirn
	Mass and Momentum > Relative Pressure	0 [Pa]
	Flow Direction > Option	Normal to Boundary Condition
	Turbulence > Option	Intensity and Length Scale
	Turbulence > Fractional Intensity	0.05
	Turbulence > Eddy Length Scale	0.25 [m]
	Additional Variables > smoke > Option	Value

Tab	Setting	Value
	Additional Variables > smoke > Add. Var. Value	0 [kg m^-3]

5. Click **OK**.

6.5.5.3. Inlet for the Vent

You will create the vent inlet boundary with a normal velocity of 0.01 m/s as prescribed in the problem description and no smoke release. The turbulence level for the inlet vent will be determined from turbulence intensity and eddy viscosity ratio.

1. Select **Insert** > **Boundary** from the menu bar or click *Boundary*  .
2. Set **Name** to `Vent`.
3. Click **OK**.
4. Apply the following settings

Tab	Setting	Value
Basic Settings	Boundary Type	Inlet
	Location	Vent
Boundary Details	Mass And Momentum > Normal Speed	0.01 [m s^-1]
	Turbulence > Option	Intensity and Eddy Viscosity Ratio
	Turbulence > Fractional Intensity	0.05
	Turbulence > Eddy Viscosity Ratio	10
	Additional Variables > smoke > Option	Value
	Additional Variables > smoke > Add. Var. Value	0 [kg m^-3]

5. Click **OK**.

6.5.6. Setting Initial Values

For this tutorial, the automatic initial values are suitable. Review and apply the default settings:

1. Click *Global Initialization*  .
2. Review the settings for velocity, pressure, turbulence and the smoke.
3. Click **OK**.

6.5.7. Setting Solver Control

CFX-Solver has the ability to calculate physical time step size for steady-state problems. If you do not know the time step size to set for your problem, you can use the `Auto Timescale` option.

1. Click *Solver Control*  .
2. Apply the following settings

Tab	Setting	Value
Basic Settings	Convergence Control > Max. Iterations	75

3. Note that **Convergence Control** > **Fluid Timescale Control** > **Timescale Control** is set to `Auto Timescale`.
4. Click **OK**.

6.5.8. Creating a Transient Flow Analysis

In this part of the tutorial, you will duplicate the steady-state analysis and adapt it to set up a transient flow analysis in CFX-Pre.

1. From the **Outline** tree view, right-click `Simulation` > `Steady State Analysis` and select **Duplicate**.
2. Right-click `Simulation` > `Copy of Steady State Analysis` and select **Rename**.
3. Rename the analysis to `Transient Analysis`.

6.5.9. Modifying the Analysis Type

In this step you will set the new analysis to a type of transient. Later, you will set the concentration of smoke for the transient analysis to rise asymptotically to its final concentration with time, so it is necessary to ensure that the interval between the time steps is smaller at the beginning of the simulation than at the end.

1. In the **Outline** tree view, ensure that `Simulation` > `Transient Analysis` is expanded.
2. From the **Outline** tree view, right-click `Simulation` > `Transient Analysis` > `Analysis Type` and select **Edit**.
3. Apply the following settings

Tab	Setting	Value
Basic Settings	Analysis Type > Option	Transient
	Analysis Type > Time Duration > Total Time	30 [s]
	Analysis Type > Time Steps > Timesteps [1 (p. 100), 2 (p. 100)]	4*0.25, 2*0.5, 2*1.0, 13*2.0 [s]
	Analysis Type > Initial Time > Option	Value
	Analysis Type > Initial Time > Time	0 [s]
1. Do *not* click *Enter Expression*  to enter lists of values. Enter the list without the units, then set the units in the drop-down list. 2. This list specifies 4 timesteps of 0.25 [s], then 2 timesteps of 0.5 [s], and so on.		

4. Click **OK**.

6.5.10. Modifying the Boundary Conditions

The only boundary condition that needs altering for the transient analysis is the `Vent` boundary condition. In the steady-state calculation, this boundary had a small amount of air flowing through it. In the transient calculation, more air passes through the vent and there is a time-dependent concentration of smoke in the air. This is initially zero, but builds up to a larger value. The smoke concentration will be specified using the CFX Expression Language.

6.5.10.1. To Modify the Vent Inlet Boundary Condition

1. In the **Outline** tree view, ensure that `Simulation` > `Transient Analysis` > `CircVent` is expanded.
2. Right-click `Simulation` > `Transient Analysis` > `CircVent` > `Vent` and select **Edit**.
3. Apply the following settings:

Tab	Setting	Value
Boundary Details	Mass And Momentum > Normal Speed	0.2 [m s^-1]

Leave the **Vent** details view open for now.

You are going to create an expression for smoke concentration. The concentration is zero for time t=0 and builds up to a maximum of 1 kg m^-3.

4. Create a new expression by selecting **Insert** > **Expressions, Functions and Variables** > **Expression** from the menu bar. Set the name to `TimeConstant`.
5. Apply the following settings

Name	Definition
TimeConstant	3 [s]

6. Click **Apply** to create the expression.
7. Create the following expressions with specific settings, remembering to click **Apply** after each is defined:

Name	Definition
FinalConcentration	1 [kg m^-3]
ExpFunction [1 (p. 101)]	FinalConcentration*abs(1-exp(-t/TimeConstant))

1. When entering this function, you can select most of the required items by right-clicking in the **Definition** window in the **Expression** details view instead of typing them. The names of the existing expressions are under the **Expressions** menu. The **exp** and **abs** functions are under **Functions > CEL**. The variable **t** is under **Variables**.

Note

The `abs` function takes the modulus (or magnitude) of its argument. Even though the expression (1- exp (-t/TimeConstant)) can never be less than zero, the abs function is included to ensure that the numerical error in evaluating it near to zero will never make the expression evaluate to a negative number.

Next, you will visualize how the concentration of smoke issued from the vent varies with time.

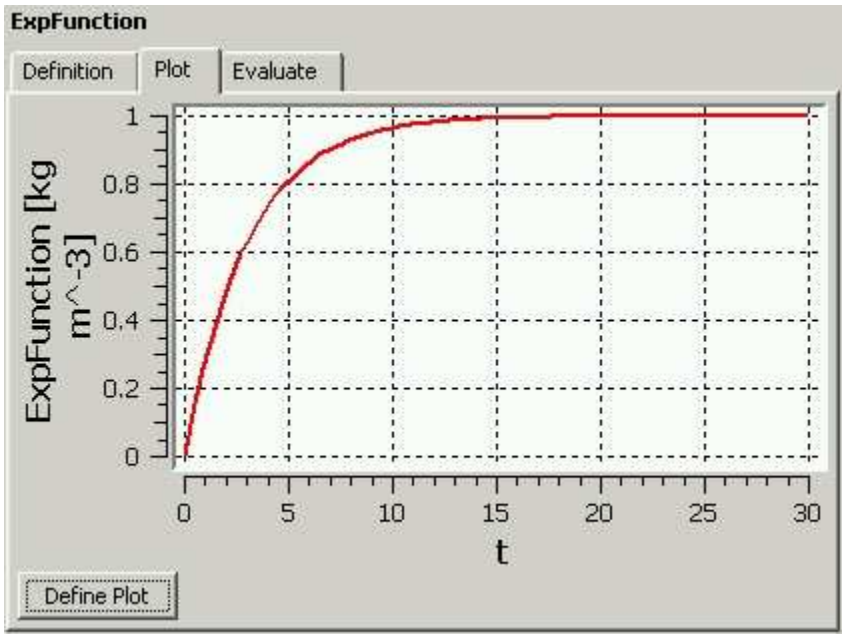
6.5.10.2. Plotting Smoke Concentration

1. Double-click `ExpFunction` in the **Expressions** tree view.
2. Apply the following settings

Tab	Setting	Value
Plot	t	(Selected)
	t > Start of Range	0 [s]
	t > End of Range	30 [s]

3. Click **Plot Expression**.

   The button name then changes to **Define Plot**, as shown.



   As can be seen, the smoke concentration rises to its asymptotic value reaching 90% of its final value at around 7 seconds.

4. Click the **Boundary: Vent** tab.

   In the next step, you will apply the expression `ExpFunction` to the additional variable `smoke` as it applies to the `Transient Analysis` boundary `Vent`.

5. Apply the following settings

Tab	Setting	Value
Boundary Details	Additional Variables > smoke > Option	Value
	Additional Variables > smoke > Add. Var. Value [1 (p. 103)]	ExpFunction
1. Click *Enter Expression*  to enter text.		

6. Click **OK**.

6.5.11. Initialization Values

When the `Transient Analysis` is run, the initial values to the CFX-Solver will be taken from the results of the `Steady State Analysis`. The steady state and transient analyses will be sequenced by setting up the configurations of these analyses in a subsequent step. For the moment, you can leave all of the initialization data set for the `Transient Analysis` to **Automatic** and the initial values will be read automatically from the `Steady State Analysis` results. Therefore, there is no need to revisit the initialization settings.

6.5.12. Modifying the Solver Control

1. In the **Outline** tree view, ensure that `Simulation` > `Transient Analysis` > `Solver` is expanded.
2. Right-click `Simulation` > `Transient Analysis` > `Solver` > `Solver Control` and select **Edit**.
3. Set **Convergence Control** > **Max. Coeff. Loops** to 3.
4. Leave the other settings at their default values.
5. Click **OK** to set the solver control parameters.

6.5.13. Output Control

To allow results to be viewed at different time steps, it is necessary to create transient results files at specified times. The transient results files do not have to contain all solution data. In this step, you will create minimal transient results files.

6.5.13.1. To Create Minimal Transient Results Files

1. In the **Outline** tree view, double click `Simulation` > `Transient Analysis` > `Solver` > `Output Control`.
2. Click the **Trn Results** tab.
3. In the **Transient Results** tree view, click *Add new item*  , set **Name** to `Transient Results 1`, and click **OK**.
4. Apply the following settings to `Transient Results 1`:

Setting	Value
Option	Selected Variables
Output Variables List [1 (p. 104)]	Pressure, Velocity, smoke

Setting	Value
Output Frequency > Option	Time List
Output Frequency > Time List [2 (p. 104)]	1, 2, 3 [s]

1. Click the *Ellipsis* ... icon to select items if they do not appear in the drop-down list. Use the **Ctrl** key to select multiple items.
2. Do *not* click *Enter Expression* to enter lists of values. Enter the list without the units, then set the units in the drop-down list.

5. Click **Apply**.
6. In the **Transient Results** tree view, click *Add new item* set **Name** to `Transient Results 2`, and click **OK**.

   This creates a second transient results object. Each object can result in the production of many transient results files.
7. Apply the following settings to `Transient Results 2`:

Setting	Value
Option	Selected Variables
Output Variables List	Pressure, Velocity, smoke
Output Frequency > Option	Time Interval
Output Frequency > Time Interval [1 (p. 104)]	4 [s]

1. A transient results file will be produced every 4 s (including 0 s) and at 1 s, 2 s and 3 s. The files will contain no mesh, and data for only the three selected variables. This reduces the size of the minimal results files. A full results file is always written at the end of the run.

8. Click **OK**.

6.5.14. Configuring Simulation Control

With two types of analysis for this simulation, configuration control is used to sequence these analyses.

6.5.14.1. Configuration Control for the Steady State Analysis

To set up the `Steady State Analysis` so that it will start at the beginning of the simulation:

1. In the **Outline** tree view, ensure that `Simulation Control` is expanded.
2. Right-click `Simulation Control` > `Configurations` and select **Insert** > **Configuration**.
3. Set **Name** to `Steady State`.
4. Click **OK**.
5. Apply the following settings

Tab	Setting	Value
General Settings	Flow Analysis	Steady State Analysis
	Activation Conditions > Activation Condition 1 > Option	Start of Simulation

6. Click **OK**.

6.5.14.2. Configuration Control for the Transient Analysis

To set up the `Transient Analysis` so that it will start upon the completion of the `Steady State Analysis`:

1. Right-click `Simulation Control` > `Configurations` and select **Insert** > **Configuration**.
2. Set **Name** to `Transient`.
3. Click **OK**.
4. Apply the following settings

Tab	Setting	Value
General Settings	Flow Analysis	Transient Analysis
	Activation Conditions > Activation Condition 1 > Option	End of Configuration
	Activation Conditions > Activation Condition 1 > Configuration Name	Steady State
Run Definition	Configuration Execution Control	Selected
	Configuration Execution Control > Initial Values Specification	Selected
	Configuration Execution Control > Initial Values Specification > Initial Values > Initial Values 1 > Option	Configuration Results
	Configuration Execution Control > Initial Values Specification > Initial Values > Initial Values 1 > Configuration Name	Steady State

5. Click **OK**.

6.5.15. Writing the CFX-Solver Input File

1. From the **Outline** tree view, right-click `Simulation Control` and select **Write Solver Input File**.
2. Apply the following settings:

Setting	Value
File of type	CFX-Solver Input Files (*.mdef)
File name	CircVent.mdef

3. Click **Save**.

This will create `CircVent.mdef` as well as a directory named `CircVent` that contains `SteadyState.cfg` and `Transient.cfg`.

4. Quit CFX-Pre, saving the case (`.cfx`) file.

6.6. Obtaining Solutions to the Steady-State and Transient Configurations

You can obtain a solution to the steady-state and transient configurations by using the following procedure.

1. Start CFX-Solver Manager.
2. From the menu bar, select **File** > **Define Run**.
3. Apply the following settings

Setting	Value
Solver Input File	CircVent.mdef
Edit Configuration	Global Settings

4. Click **Start Run**.

   CFX-Solver Manager will start with the solution of the steady-state configuration.

5. In the **Workspace** drop-down menu, select `SteadyState_001`. The residual plots for six equations will appear: U - Mom, V - Mom, W - Mom, P - Mass, K-TurbKE, and E-Diss.K (the three momentum conservation equations, the mass conservation equation and equations for the turbulence kinetic energy and turbulence eddy dissipation). The **Momentum and Mass** tab contains four of the plots and the other two are under **Turbulence Quantities**. The residual for the smoke equation is also plotted but registers no values since it is not initialized.
6. Upon the successful completion of the steady-state configuration, the solution of the transient configuration starts automatically. Notice that the text output generated by the CFX-Solver in the `Run Transient 001` **Workspace** will be more than you have seen for steady-state problems. This is because each timestep consists of several inner (coefficient) iterations. At the end of each timestep, information about various quantities is printed to the text output area. The residual for the smoke equation is now plotted under the **Additional Variables** tab.
7. Upon the successful completion of the combined steady-state and transient configurations, ensure that the check box beside **Post-Process Results** is cleared and click **OK** to close the message indicating the successful completion of the simulation.
8. In the CFX-Solver Manager, set **Workspace** to `Run CircVent 001`.
9. From the menu bar, select **Tools** > **Post-Process Results**.
10. On the **Start CFD-Post** dialog box, select **Shut down CFX-Solver Manager** and click **OK**.

6.7. Viewing the Results in CFD-Post

In this part of the tutorial, you will:

- Create an isosurface to illustrate the pattern of smoke concentration.
- View results at different time steps.
- Animate the results to view the dispersion of smoke from the vent over time.
- Save the animation as an MPEG file.

- Use volume rendering to show the visibility of smoke with its transparency.

6.7.1. Displaying Smoke Density Using an Isosurface

An isosurface is a surface of constant value of a variable. For instance, it could be a surface consisting of all points where the velocity is 1 [m s^-1]. In this case, you are going to create an isosurface of `smoke` concentration (`smoke` is the Additional Variable that you specified earlier).

1. In CFD-Post, right-click a blank area in the viewer and select **Predefined Camera** > **Isometric View (Z up)**.

   This ensures that the view is set to a position that is best suited to display the results.
2. From the menu bar, select **Insert** > **Location** > **Isosurface**, or under **Location** on the toolbar, click **Isosurface**.
3. Click **OK** to accept the default name.
4. Apply the following settings:

Tab	Setting	Value
Geometry	Variable	smoke
	Value	0.005 [kg m^-3]

5. Click **Apply**.
   - A bumpy surface is displayed, showing the smoke emanating from the vent.
   - The surface is rough because the mesh is coarse. For a smoother surface, you would re-run the problem with a smaller mesh length scale.
   - The surface will be a constant color because the default settings on the **Color** tab were used.
   - When **Color Mode** is set to either `Constant` or `Use Plot Variable` for an isosurface, the isosurface is displayed in one color.
6. In **Geometry**, experiment by changing the **Value** setting so that you can see the shape of the plume more clearly.

   Zoom in and rotate the geometry, as required.
7. When you have finished, set **Value** to `0.002 [kg m^-3]`.
8. Right-click a blank spot in the viewer and select **Predefined Camera** > **Isometric View (Z up)**.

6.7.2. Viewing the Results at Different Time Steps

When CFD-Post is loaded, the results that are immediately available are those at the final time step; in this case, at t = 30 s (this is nominally designated Final State). The **Timestep Selector** shows the **Configuration**, the step of the **Simulation**, the **Step** (outer loop) number, the **Time** (simulated time in seconds) of the configuration and the **Type** of results file that was saved at that time step for the configuration. You can see that **Partial** results files were saved (as requested in CFX-Pre) for all time steps in the transient configuration except for the last one.

1. Click *Timestep Selector*  .

2. Load the results for a **Time** value of 2 s by double-clicking the appropriate row in the **Timestep Selector**.

   After a short pause, the **Current Timestep** (located just below the title bar of the **Timestep Selector**) will be updated with the new time step number.

3. Load the time value of 4 s using the **Timestep Selector**.

   The smoke has now spread out even more, and is being carried by the wind.

4. Double-click some more time values to see how the smoke plume grows with time.

5. Finish by loading a time value of 12 s.

6.7.3. Generating Titled Image Files

You can produce titled image output from CFD-Post.

6.7.3.1. Adding a Title

First, you will add text to the viewer so that the printed output has a title.

1. Select **Insert** > **Text** from the menu bar or click *Create text*  .

2. Click **OK** to accept default name.

3. In the **Text String** box, enter the following text.

```
Isosurface showing smoke concentration of 0.002 kg/m^3 after
```

Note

Further text will be added at a later stage to complete this title.

4. Select **Embed Auto Annotation**.

5. Set **Type** to `Time Value`.

   In the text line, note that <aa> has been added to the end. This is where the time value will be placed.

6. Click **Apply** to create the title.

7. Click the **Location** tab to modify the position of the title.

   The default settings for text objects center text at the top of the screen. To experiment with the position of the text, change the settings on the **Location** tab.

8. Under the **Appearance** tab, change **Color Mode** to `User Specified` and select a new color.

9. Click **Apply**.

6.7.3.2. JPEG output

CFD-Post can save images in several different formats. In this section you will save an image in JPEG format.

1. Select **File** > **Save Picture**, or click *Save Picture*  .

2. Set **Format** to `JPEG`.

3. Click *Browse*  next to the **File** data box.
4. Browse to the directory where you want the file saved.
5. Enter a name for the JPEG file.
6. Click **Save** to set the file name and directory.

   This sets the path and name for the file.
7. To save the file, click **Save** on the **Save Picture** dialog box.

   To view the file or make a hard copy, use an application that supports JPEG files.
8. Turn off the visibility of the text object to hide it.

6.7.4. Generating a Movie

You can generate an MPEG file to show the transient flow of the plume of smoke. To generate a movie file, you use the **Animation** dialog box.

1. In the **Timestep Selector**, ensure that a time value of 0 s is loaded.
2. Click *Animation*  .
3. Ensure that **Quick Animation** is selected.
4. Position the geometry so that you will be able to see the plume of smoke.
5. In the object tree of the **Animation** dialog box, click **Timesteps**.
6. Click *More Animation Options*  to show more animation settings.
7. Ensure that the *Repeat forever* button  next to **Repeat** is not selected (not depressed).
8. Select **Save Movie**.
9. Set **Format** to `MPEG1`.
10. Click *Browse*  next to **Save Movie**.
11. Set **File name** to `CircVent.mpg`.
12. If required, set the path location to a different directory.
13. Click **Save**.

    The movie file name (including path) has been set, but the animation has not yet been produced.
14. Click *Play the animation*  .

    - The movie will be created as the animation proceeds.
    - This will be slow, because for each timestep results will be loaded and objects will be created.
    - To view the movie file, you need to use a viewer that supports the MPEG format.

Note

To explore additional animation options, click the **Options** button. On the **Advanced** tab of **Animation Options**, there is a check box called **Save Frames As Image Files**. By selecting this check box, the JPEG or PPM files used to encode each frame of the movie will persist after movie creation; otherwise, they will be deleted.

6.7.5. Viewing the Dispersion of Smoke at the Final Time Step

The final time step has the greatest dispersion of smoke, so you will load only that time step, then view the smoke using the Volume Rendering feature.

1. Select **File** > **Load Results**.
2. In the **Load Results File** dialog box, select **Load only the last results** (the other default settings should remain unchanged) and ensure that **File name** is set to **CircVent_001.mres**. Click **Open**.

   Note

   A warning message appears. Click **OK**.

3. If necessary, right-click a blank spot in the viewer and select **Predefined Camera** > **Isometric View (Z up)**.
4. In the **Outline** view, clear **Isosurface 1** and **Text 1**.
5. Select **Insert** > **Volume Rendering** and set the **Name** to be `SmokeVolume`.
6. In the **Details** view, set the following values:

Tab	Setting	Value
Geometry	Variable	smoke
	Resolution	50
	Transparency	.2
Color	Mode	Variable
	Variable	smoke
	Color Map	Greyscale

   Click **Apply**.
7. When you have finished, quit CFD-Post.

Chapter 7: Flow Around a Blunt Body

This tutorial includes:

7.1. Tutorial Features

In this tutorial you will learn about:

- Solving and post-processing a case where the geometry has been omitted on one side of a symmetry plane.
- Using free-slip wall boundaries as a compromise between accurate flow modeling and computational grid size.
- Accurately modeling the near-wall flow using Shear Stress Transport (SST) turbulence model.
- Running the CFX-Solver in parallel (optional).
- Creating vector plots in CFD-Post with uniform spacing between the vectors.
- Creating a macro using power syntax in CFD-Post.

Component	Feature	Details
CFX-Pre	User Mode	General Mode
	Analysis Type	Steady State
	Fluid Type	Ideal Gas
	Domain Type	Single Domain
	Turbulence Model	Shear Stress Transport
	Heat Transfer	Isothermal
	Boundary Conditions	Inlet (Subsonic)
		Outlet (Subsonic)
		Symmetry Plane
		Wall: No-Slip
		Wall: Free-Slip
	Timestep	Physical Time Scale
CFX-Solver Manager	Parallel processing	
CFD-Post	Plots	Default Locators
		Outline Plot (Wireframe)

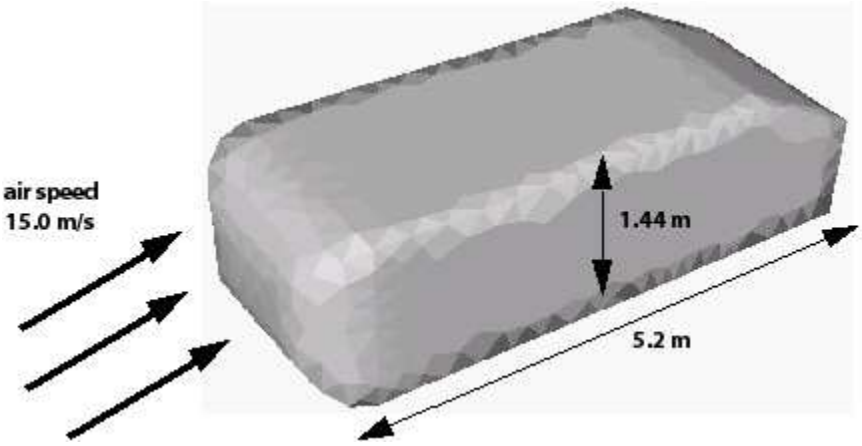
Component	Feature	Details
		Sampling Plane
		Streamline
		Vector
		Volume
	Other	Changing the Color Range
		Instancing Transformation
		Lighting Adjustment
		Symmetry
		Viewing the Mesh

7.2. Overview of the Problem to Solve

In this tutorial, a generic vehicle body is placed into an oncoming side wind of 15 m/s. The turbulence will be set to Intensity and Length scale with a value of 0.05, which corresponds to 5% turbulence, a medium level intensity, and with an Eddy Length scale value of 0.1 m.

The goal of this tutorial is to accurately model the behavior of the flow around the body. Since both the geometry and the flow are symmetric about a vertical plane, only half of the geometry will be used to find the CFD solution. The overall approach to solving this problem is to first set the free-slip wall boundaries on the sides of and above the domain. The near-wall flow will then be modeled using Shear Stress Transport. Vector plots will finally be created to display the near wall flow behavior.

Figure 7.1 External Air Flow Over a Generic Vehicle Body



7.3. Before You Begin

If this is the first tutorial you are working with, it is important to review the following topics before beginning:

- *Preparing a Working Directory and Starting ANSYS CFX in Standalone Mode* (p. 1)
- *Running ANSYS CFX Tutorials in ANSYS Workbench* (p. 2)
- *Changing the Display Colors* (p. 5)
- *Playing a Tutorial Session File* (p. 4)

7.4. Starting CFX-Pre

1. Prepare the working directory using the following files in the `examples` directory:
   - `BluntBody.pre`
   - `BluntBodyDist.cse`
   - `BluntBodyMesh.gtm`
2. Set the working directory and start CFX-Pre.

   For details, see *Preparing a Working Directory and Starting ANSYS CFX in Standalone Mode* (p. 1)

7.5. Defining a Case in CFX-Pre

If you want to set up the simulation automatically using a tutorial session file, run `BluntBody.pre`. For details, see *Playing a Tutorial Session File* (p. 4). Then proceed to *Obtaining a Solution Using CFX-Solver Manager* (p. 118).

If you want to set up the simulation manually, proceed to the following steps:

1. In CFX-Pre, select **File** > **New Case**.
2. Select **General** and click **OK**.
3. Select **File** > **Save Case As**.
4. Under **File name**, type `BluntBody`.
5. Click **Save**.

7.5.1. Importing the Mesh

1. Edit `Case Options` > `General` in the **Outline** tree view and ensure that **Automatic Default Domain** is turned off.

   Default Domain generation should be turned off because you will create a new domain manually, later in this tutorial.
2. Right-click `Mesh` and select **Import Mesh** > **Other**. The **Import Mesh** dialog box appears.
3. Apply the following settings:

Setting	Value
File name	BluntBodyMesh.gtm

4. Click **Open**.

7.5.2. Creating the Domain

The flow of air in the domain is expected to be turbulent and approximately isothermal at 288 K. The Shear Stress Transport (SST) turbulence model with automatic wall function treatment will be used because of its highly accurate predictions of flow separation. To take advantage of the SST model, the boundary layer should be resolved with at least 10 mesh nodes. In order to reduce computational time, the mesh in this tutorial is much coarser than that.

This tutorial uses air as an ideal gas as the fluid. When modeling a compressible flow using the ideal gas approximation to calculate density variations, it is important to set a realistic reference pressure. This is because some fluid properties depend on the absolute fluid pressure (calculated as the static pressure plus the reference pressure which is chosen to be 1 atm).

Tip

For more details, see The Shear Stress Transport (SST).

1. Ensure that `Flow Analysis 1` > `Default Domain` is deleted. If not, right-click `Default Domain` and select **Delete**.
2. Click *Domain*  , and set the name to `BluntBody`.
3. Apply the following settings to `BluntBody`:

Tab	Setting	Value
Basic Settings	Location and Type > Location	B1.P3
	Fluid and Particle Definitions	Fluid 1
	Fluid and Particle Definitions > Fluid 1 > Material	Air Ideal Gas
	Domain Models > Pressure > Reference Pressure	1 [atm]
Fluid Models	Heat Transfer > Option	Isothermal
	Heat Transfer > Fluid Temperature	288 [K]
	Turbulence > Option	Shear Stress Transport

4. Click **OK**.

7.5.3. Creating Composite Regions

An imported mesh may contain many 2D regions. For the purpose of creating boundary conditions, it can sometimes be useful to group several 2D regions together and apply a single boundary to the composite 2D region. In this case, you are going to create a **Union** between two regions that both require a free-slip wall boundary.

1. From the main menu, select **Insert** > **Regions** > **Composite Region**.
2. Set the name to `FreeWalls` and click **OK**.
3. Apply the following settings:

Tab	Setting	Value
Basic Settings	Dimension (Filter)	2D

4. Click  beside the **Region List** dialog box, to display the **Selection Dialog**. Hold down the **Ctrl** key and select `Free1` and `Free2`.
5. Click **OK** to confirm your selection.

6. Click **OK** to create the composite region.

7.5.4. Creating the Boundaries

The simulation requires inlet, outlet, wall (no slip and free-slip) and symmetry plane boundaries. The regions for these boundaries were defined when the mesh was created (except for the composite region just created for the free-slip wall boundary).

7.5.4.1. Inlet Boundary

1. Click *Boundary* .
2. Under **Name**, type `Inlet`.
3. Apply the following settings:

Tab	Setting	Value
Basic Settings	Boundary Type	Inlet
	Location	Inlet
Boundary Details	Flow Regime > Option	Subsonic
	Mass and Momentum > Option	Normal Speed
	Mass and Momentum > Normal Speed	15 [m s^-1]
	Turbulence > Option	Intensity and Length Scale
	Turbulence > Fractional Intensity	0.05[a]
	Turbulence > Eddy Length Scale	0.1 [m][a]

[a]From the problem description.

4. Click **OK**.

7.5.4.2. Outlet Boundary

1. Create a new boundary named `Outlet`.
2. Apply the following settings:

Tab	Setting	Value
Basic Settings	Boundary Type	Outlet
	Location	Outlet
Boundary Details	Mass and Momentum > Option	Static Pressure
	Mass and Momentum > Relative Pressure	0 [Pa]

3. Click **OK**.

7.5.4.3. Free-Slip Wall Boundary

The top and side surfaces of the rectangular region will use free-slip wall boundaries.

- On free-slip walls the shear stress is set to zero so that the fluid is not retarded.
- The velocity normal to the wall is also set to zero.
- The velocity parallel to the wall is calculated during the solution.

This boundary is an approximation that may not accurately represent the true flow conditions. By using a free-slip wall boundary, the flow modeling will be less accurate but the computational grid size can be reduced by modeling less of the surroundings. If this case were modeling a wind tunnel experiment, the geometry would match the size and shape of the wind tunnel and use no-slip walls. If this case were modeling a blunt body open to the atmosphere, a much larger domain would be used to minimize the effect of the far-field boundary, and the far-field boundary type would be set to either a free-slip wall or a pressure-specified entrainment opening.

You will apply a single boundary to both walls by using the composite region defined earlier.

1. Create a new boundary named `FreeWalls`.
2. Apply the following settings:

Tab	Setting	Value
Basic Settings	Boundary Type	Wall
	Location	FreeWalls
Boundary Details	Mass and Momentum > Option	Free Slip Wall

3. Click **OK**.

7.5.4.4. Symmetry Plane Boundary

1. Create a new boundary named SymP.
2. Apply the following settings:

Tab	Setting	Value
Basic Settings	Boundary Type	Symmetry
	Location	SymP

3. Click **OK**.

7.5.4.5. Wall Boundary on the Blunt Body Surface

1. Create a new boundary named Body.
2. Apply the following settings:

Tab	Setting	Value
Basic Settings	Boundary Type	Wall
	Location	Body
Boundary Details	Mass and Momentum > Option	No Slip Wall

3. Click **OK**.

The remaining 2D regions (in this case, just the low Z face) will be assigned the default boundary which is an adiabatic, no-slip wall condition. In this case, the name of the default boundary is **BluntBody Default**. Although the boundaries **Body** and **BluntBody Default** are identical (except for their locations), the **Body** boundary was created so that, during post-processing, its location can be conveniently distinguished from the other adiabatic, no-slip wall surfaces.

7.5.5. Setting Initial Values

The initial conditions are consistent with inlet boundaries.

1. Click *Global Initialization*  .
2. Apply the following settings:

Tab	Setting	Value
Global Settings	Initial Conditions > Cartesian Velocity Components > Option	Automatic with Value
	Initial Conditions > Cartesian Velocity Components > U	15 [m s^-1]
	Initial Conditions > Cartesian Velocity Components > V	0 [m s^-1]
	Initial Conditions > Cartesian Velocity Components > W	0 [m s^-1]

3. Click **OK**.

7.5.6. Setting Solver Control

1. Click *Solver Control*  .
2. Apply the following settings:

Tab	Setting	Value
Basic Settings	Convergence Control > Max. Iterations	60
	Convergence Control > Fluid Timescale Control > Timescale Control	Physical Timescale
	Convergence Control > Fluid Timescale Control > Physical Timescale	2 [s][a]
	Convergence Criteria > Residual Target	1e-05

[a]Based on the air speed and the size of the object.

3. Click **OK**.

7.5.7. Writing the CFX-Solver Input (.def) File

1. Click *Define Run*  .
2. Apply the following settings:

Setting	Value
File name	BluntBody.def

3. Click **Save**.

   CFX-Solver Manager automatically starts and, on the **Define Run** dialog box, the **Solver Input File** is set.

4. If using Standalone Mode, quit CFX-Pre, saving the simulation (`.cfx`) file at your discretion.

7.6. Obtaining a Solution Using CFX-Solver Manager

This tutorial introduces the parallel solver capabilities of CFX.

Note

The results produced will be identical, whether produced by a parallel or serial run.

If you do not want to solve this tutorial in parallel (on more than one processor) or you do not have a license to run the CFX-Solver in parallel, proceed to *Obtaining a Solution in Serial* (p. 118).

If you do not know if you have a license to run the CFX-Solver in parallel, you should either ask your system administrator, or query the license server (see the ANSYS, Inc. Licensing Guide (which is installed with the ANSYS License Manager) for details). Alternatively proceed to *Obtaining a Solution in Serial* (p. 118).

If you would like to solve this tutorial in parallel on the same machine, proceed to *Obtaining a Solution with Local Parallel* (p. 119).

If you would like to solve this tutorial in parallel across different machines, proceed to *Obtaining a Solution with Distributed Parallel* (p. 120).

7.6.1. Obtaining a Solution in Serial

When CFX-Solver Manager has started, you can obtain a solution to the CFD problem by using the following procedure.

1. Click **Start Run**.
2. When CFX-Solver is finished, select the check box next to **Post-Process Results**.
3. If using Standalone Mode, select the check box next to **Shut down CFX-Solver Manager**.
4. Click **OK**. Continue this tutorial from *Viewing the Results in CFD-Post* (p. 122).

7.6.2. Obtaining a Solution in Parallel

7.6.2.1. Background to Parallel Running in CFX

Using the parallel capability of the CFX-Solver allows you to divide a large CFD problem so that it can run on more than one processor/machine at once. This saves time and, when multiple machines are used, avoids problems which arise when a CFD calculation requires more memory than a single machine has available. The partition (division) of the CFD problem is automatic.

A number of events occur when you set up a parallel run and then ask the CFX-Solver to calculate the solution:

- Your mesh will be divided into the number of partitions that you have chosen.
- The CFX-Solver runs separately on each of the partitions on the selected machine(s).
- The results that one CFX-Solver process calculates affects the other CFX-Solver processes at the interface between the different sections of the mesh.
- All of the CFX-Solver processes are required to communicate with each other and this is handled by the *master* process.
- The master process always runs on the machine that you are logged into when the parallel run starts. The other CFX-Solver processes are *slave* processes and may be run on other machines.
- After the problem has been solved, a single results file is written. It will be identical to a results file from the same problem run as a serial process, with one exception: an extra variable `Real partition number` will be available for the parallel run. This variable will be used later in this tutorial during post processing.

7.6.2.2. Obtaining a Solution with Local Parallel

To run in local parallel mode, the machine you are on must have more than one processor.

In CFX-Solver Manager, the **Define Run** dialog box should already be open.

1. Leave **Type of Run** set to `Full`.

   If **Type of Run** was instead set to `Partitioner Only`, your mesh would be split into a number of partitions but would not be run in the CFX-Solver afterwards.
2. Set **Run Mode** to a parallel mode suitable for your configuration; for example, `HP MPI Local Parallel`.

   This is the recommended method for most applications.
3. If required, click *Add Partition*  to add more partitions.

   By default, 2 partitions are assigned.
4. Select **Show Advanced Controls**.
5. Click the **Partitioner** tab at the top of the dialog box.
6. Use the default `MeTiS` partitioner.

   Your model will be divided into two sections, with each section running in its own CFX-Solver process. The default is the `MeTiS` partitioner because it produces more efficient partitions than either `Recursive Coordinate Bisection` or `User Specified Direction`.
7. Click **Start Run**.
8. When CFX-Solver is finished, select the check box next to **Post-Process Results**.
9. If using Standalone Mode, select the check box next to **Shut down CFX-Solver Manager**.
10. Click **OK**.

Continue this tutorial from *Text Output when Running in Parallel* (p. 121).

7.6.2.3. Obtaining a Solution with Distributed Parallel

Before running in Distributed Parallel mode, please ensure that your system has been configured as described in the installation documentation.

In CFX-Solver Manager, the **Define Run** dialog box should already be open.

1. Leave **Type of Run** set to `Full`.

   If **Type of Run** was instead set to `Partitioner Only`, your mesh would be split into a number of partitions but would not be run in the CFX-Solver afterwards.
2. Set **Run Mode** to a parallel mode suitable for your environment; for example, `HP MPI Distributed Parallel`.

   The name of the machine that you are currently logged into should be in the **Host Name** list. You are going to run with two partitions on two different machines, so another machine must be added.
3. Click *Insert Host*  to specify a new host machine.

   - The **Select Parallel Hosts** dialog box is displayed. This is where you choose additional machines to run your processes.
   - Your system administrator should have set up a hosts file containing a list of the machines that are available to run the parallel CFX-Solver.
   - The **Host Name** column displays names of available hosts.
   - The second column shows the number of processors on that machine.
   - The third shows the relative processor speed: a processor on a machine with a relative speed of 1 would typically be twice as fast as a machine with a relative speed of 0.5.
   - The last column displays operating system information.
   - This information is read from the hosts file; if any information is missing or incorrect your system administrator should correct the hosts file.

   Note

   The # processors, relative speed and system information does not have to be specified to be able to run on a host.
4. Select the name of another machine in the **Host Name** list.

   Select a machine that you can log into.
5. Click **Add**.

   The name of the machine is added to the **Host Name** column.

   Note

   Ensure that the machine that you are currently logged into is in the **Hosts Name** list in the **Define Run** dialog box.

6. **Close** the **Select Parallel Hosts** dialog box.
7. Select **Show Advanced Controls**.
8. Click the **Partitioner** tab at the top of the dialog box.
9. Use the default `MeTiS` partitioner.

   Your model will be divided into two sections, with each section running in its own CFX-Solver process. The default is the `MeTiS` partitioner because it produces more efficient partitions than either `Recursive Coordinate Bisection` or `User Specified Direction`.
10. Click **Start Run** to begin the parallel run.
11. Click **OK** on the pop-up message.
12. When CFX-Solver is finished, select the check box next to **Post-Process Results**.
13. If using Standalone Mode, select the check box next to **Shut down CFX-Solver Manager**.
14. Click **OK**.

7.6.2.4. Text Output when Running in Parallel

The text output area shows what is being written to the output file. You will see information similar to the following:

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

Run mode:       partitioning run

Host computer:  fastmachine1
Job started:    Tue Jan 20 14:13:27 2009
```

This tells you that the information following is concerned with the partitioning. After the partitioning job has finished, you will find:

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

Partitioning information for domain: BluntBody

       +------------------+------------------------+------------------+
       |     Elements     |        Vertices        |     Faces        |
+------+------------------+------------------------+------------------+
| Part |   Number     %   |   Number     %   %Ovlp |  Number     %    |
+------+------------------+------------------------+------------------+
| Full |    131878        |     37048              |    11318         |
+------+------------------+------------------------+------------------+
|    1 |     67873   50.4 |     19431   50.4   4.0 |     5705   49.5  |
|    2 |     66865   49.6 |     19151   49.6   4.0 |     5820   50.5  |
+------+------------------+------------------------+------------------+
| Sum  |    134738  100.0 |     38582  100.0   4.0 |    11525  100.0  |
+------+------------------+------------------------+------------------+

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

- Preparations                                        3.689E-01 seconds
- Low-level mesh partitioning                        5.599E-02 seconds
- Global partitioning information                    9.998E-03 seconds
- Element and face partitioning information          7.999E-03 seconds
```

```
- Vertex partitioning information                   0.000E+00 seconds
- Partitioning information compression              0.000E+00 seconds
- Summed CPU-time for mesh partitioning             4.609E-01 seconds


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

Host computer:  fastmachine1
Job finished:   Tue Jan 20 14:13:29 2009

Total CPU time: 9.749E-01 seconds
            or: (          0:         0:         0:     0.975 )
                (       Days:     Hours:   Minutes:   Seconds )

Total wall clock time: 2.000E+00 seconds
            or: (          0:         0:         0:     2.000 )
                (       Days:     Hours:   Minutes:   Seconds )
```

This marks the end of the partitioning job. The CFX-Solver now begins to solve your parallel run:

```
+--------------------------------------------------------------------+
|                          Job Information                           |
+--------------------------------------------------------------------+
Run mode:       parallel run (HP MPI Distributed Parallel)
Host computer:  fastmachine1
Par. Process:   Master running on mesh partition:       1
Job started:    Thu Nov 28 15:19:20 2005
Host computer:  slowermachine
Par. Process:   Slave  running on mesh partition:       2
Job started:    Thu Nov 28 15:24:55 2005
```

The machine that you are logged into runs the master process, and controls the overall simulation. The second machine selected will run the slave process. If you had more than two processes, each additional process is run as a slave process.

The master process in this example is running on the mesh partition number 1 and the slave is running on partition number 2. You can find out which nodes and elements are in each partition by using CFD-Post later on in the tutorial.

When the CFX-Solver finishes, the output file displays the job information and a pop-up message to indicate completion of the run.

7.7. Viewing the Results in CFD-Post

In CFD-Post, you will create an instance transform object to recreate the full geometry. This full geometry will then be displayed using a reflection transform. You will also create a vector plot that will let you see how the flow behaves around the body as well as a pressure plot displaying the pressure distribution on the body. In order to show the path of air along the surface of the blunt body, you will make a surface streamline. Finally, the values of the dimensionless wall distance y^+ will be examined to make sure the mesh used in the simulation was fine enough near the walls.

7.7.1. Using Symmetry Plane to Display the Full Geometry

Earlier in this tutorial you used a symmetry plane boundary because the entire blunt body is symmetrical about a plane. Due to this symmetry, it was necessary to use only half of the full geometry to calculate the CFD results. However, for visualization purposes, it is helpful to use the full blunt body. CFD-Post is able to recreate the full data set from the half that was originally calculated. This is done by creating an **Instance Transform** object.

7.7.1.1. Manipulating the Geometry

You need to manipulate the geometry so that you will be able to see what happens when you use the symmetry plane. The CFD-Post features that you have used in earlier tutorials will not be described in detail. New features will be described in detail.

- Right-click a blank area in the viewer and select **Predefined Camera** > **View From +X**.

7.7.1.2. Creating an Instance Transform

Instance Transforms are used to visualize a full geometry representation in cases where the simulation took advantage of symmetry to solve for only part of the geometry. There are three types of transforms that you can use: Rotation, Translation, Reflection/Mirroring. In this tutorial, you will create a Reflection transform located on a plane.

1. Click **Location** > **Plane** and set the name to `Reflection Plane`.
2. Apply the following settings:

Tab	Setting	Value
Geometry	Definition > Method	ZX Plane
	Definition > Y	0.0 [m]
Render	Show Faces	(cleared)

3. Click **Apply**.

   This creates a plane at y=0, the same location as the symmetry plane defined in CFX-Pre. Now the instance transform can be created using this plane:
4. From the main menu, select **Insert** > **Instance Transform** and accept the default name.
5. Apply the following settings:

Tab	Setting	Value
Definition	Instancing Info From Domain	(Cleared)
	Apply Rotation	(Cleared)
	Apply Reflection	(Selected)
	Apply Reflection > Plane	Reflection Plane

6. Click **Apply**.

7.7.1.3. Using the Reflection Transform

You can apply the new transform to graphics objects. For example, you can modify the display of the wireframe as follows:

1. Under the **Outline** tab, in `User Locations and Plots`, apply the following settings to `Wireframe`:

Tab	Setting	Value
View	Apply Instancing Transform > Transform	Instance Transform 1

2. Click **Apply**.
3. Zoom so that the geometry fills the Viewer.

   You will see the full blunt body.

In this case, you created a new instance transform and applied it to the wireframe. This caused only the wireframe object to be mirrored. If you had modified the default transform instead of creating a new one, then all graphics (including those not yet made) would be mirrored by default.

7.7.2. Creating Velocity Vectors

You are now going to create a vector plot to show velocity vectors behind the blunt body. You need to first create an object to act as a locator, which, in this case, will be a sampling plane. Then, create the vector plot itself.

7.7.2.1. Creating the Sampling Plane

A sampling plane is a plane with evenly spaced sampling points on it.

1. Right-click a blank area in the viewer and select **Predefined Camera** > **View From +Y**.

   This ensures that the changes can be seen.
2. Create a new plane named `Sample`.
3. Apply the following settings to create a sampling plane that is parallel to the ZX plane and located at x= 6 m, y= 0.001 m and z= 1 m relative to blunt object:

Tab	Setting	Value
Geometry	Definition > Method	Point and Normal
	Definition > Point	6, -0.001, 1
	Definition > Normal	0, 1, 0
	Plane Bounds > Type	Rectangular
	Plane Bounds > X Size	2.5 [m]
	Plane Bounds > Y Size	2.5 [m]
	Plane Type	Sample
	Plane Type > X Samples	20
	Plane Type > Y Samples	20
Render	Show Faces	(Cleared)
	Show Mesh Lines	(Selected)

4. Click **Apply**.

   You can zoom in on the sampling plane to see the location of the sampling points (where lines intersect). There are a total of 400 (20 * 20) sampling points on the plane. A vector can be created at each sampling point.
5. Turn off the visibility of `Sample`.

7.7.2.2. Creating a Vector Plot Using Different Sampling Methods

1. Click *Vector* and accept the default name.
2. Apply the following settings:

Tab	Setting	Value
Geometry	Definition > Locations	Sample
	Definition > Sampling	Vertex
Symbol	Symbol Size	0.25

3. Click **Apply**.
4. Zoom until the vector plot is roughly the same size as the viewer.

   You should be able to see a region of recirculation behind the blunt body.
5. Ignore the vertices on the sampling plane and increase the density of the vectors by applying the following settings:

Tab	Setting	Value
Geometry	Definition > Sampling	Equally Spaced
	Definition > # of Points	1000

6. Click **Apply**.
7. Change the location of the Vector plot by applying the following setting:

Tab	Setting	Value
Geometry	Definition > Locations	SymP

8. Click **Apply**.

7.7.3. Displaying Pressure Distribution on Body and Symmetry Plane

1. Apply the following settings to the boundary named Body:

Tab	Setting	Value
Color	Mode	Variable
	Variable	Pressure
View	Apply Instancing Transform > Transform	Instance Transform 1

2. Click **Apply**.
3. Apply the following settings to SymP:

Tab	Setting	Value
Render	Show Faces	(Cleared)
	Show Mesh Lines	(Selected)

4. Click **Apply**.

   You will be able to see the mesh around the blunt body, with the mesh length scale decreasing near the body, but still coarse in the region of recirculation. By zooming in, you will be able to see the layers of inflated elements near the body.

7.7.4. Creating Surface Streamlines to Display the Path of Air along the Surface of the Body

In order to show the path of air along the surface of the blunt body, surface streamlines can be made as follows:

1. Turn off the visibility of `Body`, `SymP` and `Vector 1`.
2. Create a new plane named `Starter`.
3. Apply the following settings:

Tab	Setting	Value
Geometry	Definition > Method	YZ Plane
	X	-0.1 [m]

4. Click **Apply**.
5. Right-click a blank area in the viewer and select **Predefined Camera** > **Isometric View (Z up)**.

   The plane appears just upstream of the blunt body.
6. Turn off the visibility of the plane.

   This hides the plane from view, although the plane still exists.
7. Click *Streamline*  and click **OK** to accept the default name.
8. Apply the following settings:

Tab	Setting	Value
Geometry	Type	Surface Streamline
	Definition > Surfaces	Body
	Definition > Start From	Locations
	Definition > Locations	Starter
	Definition > Max Points	100
	Definition > Direction	Forward

9. Click **Apply**.

The surface streamlines appear on half of the surface of the blunt body. They start near the upstream end because the starting points were formed by projecting nodes from the plane to the blunt body.

7.7.5. Moving Objects

In CFD-Post, you can reposition some locator objects directly in the viewer by using the mouse.

1. Turn on the visibility of the plane named `Starter`.
2. Select the  **Single Select** mouse pointer from the **Selection Tools** toolbar.
3. In the viewer, click the `Starter` plane to select it, then use the left mouse button to drag it along the X axis.

   Notice that the streamlines are redrawn as the plane moves. The rate at which the streamlines are re-drawn is dependent on your computer's speed. If the streamlines are updated infrequently, you may find it useful to move the mouse very slowly.

7.7.6. Creating a Surface Plot of y+

The velocity next to a no-slip wall boundary changes rapidly from a value of zero at the wall to the free stream value a short distance away from the wall. This layer of high velocity gradient is known as the boundary layer. Many meshes are not fine enough near a wall to accurately resolve the velocity profile in the boundary layer. Wall functions can be used in these cases to apply an assumed functional shape of the velocity profile. Other grids are fine enough that they do not require wall functions, and application of the latter has little effect. The majority of cases fall somewhere in between these two extremes, where the boundary layer is partially resolved by nodes near the wall and wall functions are used to supplement accuracy where the nodes are not sufficiently clustered near the wall.

One indicator of the closeness of the first node to the wall is the dimensionless wall distance y^+. It is good practice to examine the values of y^+ at the end of your simulation. At the lower limit, a value of y^+ less than or equal to 11 indicates that the first node is within the laminar sublayer of the boundary flow. Values larger than this indicate that an assumed logarithmic shape of the velocity profile is being used to model the boundary layer portion between the wall and the first node. Ideally you should confirm that there are several nodes (3 or more) resolving the boundary layer profile. If this is not observed, it is highly recommended that more nodes be added near the wall surfaces in order to improve simulation accuracy. In this tutorial, a coarse mesh is used to reduce the run time. Thus, the grid is far too coarse to resolve any of the boundary layer profile, and the solution is not highly accurate.

7.7.6.1. Surface Plot of y+

A surface plot is one which colors a surface according to the values of a variable: in this case, y^+. A surface plot of y^+ can be obtained as follows:

1. Turn off the visibility of all previous plots.
2. Under the **Outline** tab, apply the following settings to `BluntBodyDefault`:

Tab	Setting	Value
Color	Mode	Variable
	Variable	Yplus [1 (p. 127)]
View	Apply Instancing Transform > Transform	Instance Transform 1

Footnote

1. Click the *Ellipsis* ... icon to the right of the **Variable** dropdown menu to view a full list of variables, including **Yplus**.

3. Click **Apply**.
4. Under the **Outline** tab, apply the following settings to Body:

Tab	Setting	Value
Color	Mode	Variable
	Variable	Yplus [1 (p. 128)]
View	Apply Instancing Transform > Transform	Instance Transform 1

Footnote

1. Click the *Ellipsis* ... icon to the right of the **Variable** dropdown menu to view a full list of variables, including **Yplus**.

5. Click **Apply**.

7.7.7. Demonstrating Power Syntax

This section demonstrates a power syntax macro used to evaluate the variation of any variable in the direction of the x-axis. This is an example of power syntax programming in CFD-Post.

A macro containing CCL and power syntax will be loaded by playing a session file. This macro will be executed by entering a line of power syntax in the **Command Editor** dialog box. The macro tells CFD-Post to create slice planes, normal to the X axis, at 20 evenly-spaced locations from the beginning to the end of the domain. On each plane, it measures and prints the minimum, maximum, and average values for a specified variable (using conservative values). The planes are colored using the specified variable.

Note

The CFD-Post engine can respond to CCL commands issued directly, or to commands issued using the graphical user interface. The **Command Editor** dialog box can be used to enter any valid CCL command directly.

1. Play the session file named `BluntBodyDist.cse`.
2. Right-click a blank area in the viewer and select **Predefined Camera** > **View From -X**.
3. Select **Tools** > **Command Editor** from the menu bar.
4. Type the following line into the **Command Editor** dialog box (the quotation marks and the semi-colon are required):

```
!BluntBodyDist("Velocity u");
```

5. Click **Process**.

   The minimum, maximum and average values of the variable at each X location are written to the file `BluntBody.txt`. The results can be viewed by opening the file in a text editor.

You can also run the macro with a different variable.

To view the content of the session file (which contains explanatory comments), open the session file in a text editor. It contains all of the CCL and power syntax commands and will provide a better understanding of how the macro works.

7.7.8. Viewing the Mesh Partitions (Parallel Only)

If you solved this tutorial in parallel, then an additional variable named `Real partition number` will be available in CFD-Post

1. Create an Isosurface of `Real partition number` equal to `1`.
2. Create a second Isosurface of `Real partition number` equal to `1.999`.

The two Isosurfaces show the edges of the two partitions. The gap between the two plots shows the overlap nodes. These were contained in both partitions 1 and 2.

When you have finished looking at the results, quit CFD-Post.

Chapter 8: Buoyant Flow in a Partitioned Cavity

This tutorial includes:

8.1. Tutorial Features

In this tutorial you will learn about:

- Using CFX-4 Mesh Import.
- Setting up a time dependent (transient) simulation.
- Modeling buoyant flow.

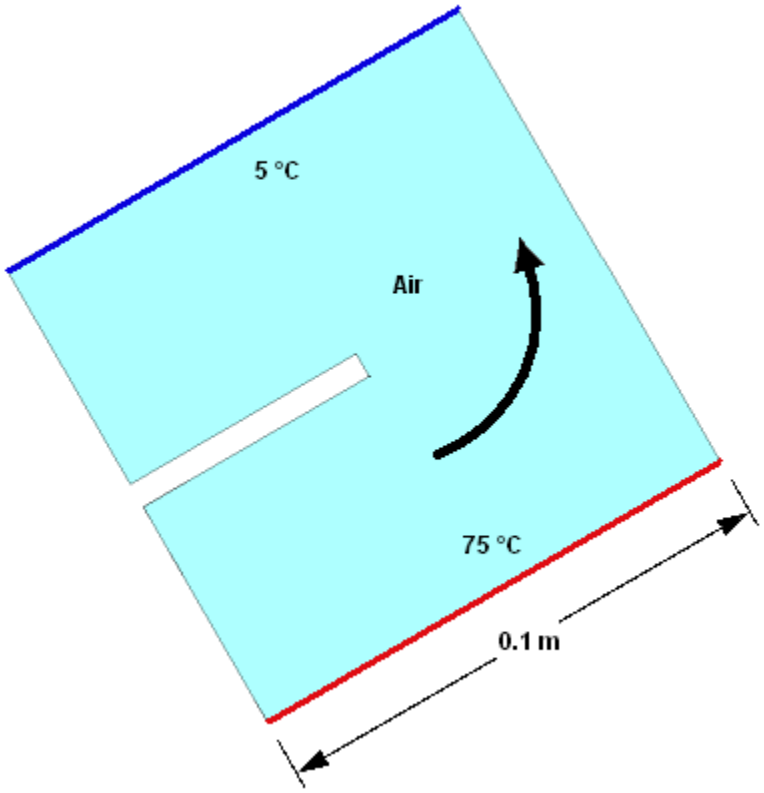
Component	Feature	Details
CFX-Pre	User Mode	General Mode
	Analysis Type	Transient
	Fluid Type	General Fluid
	Domain Type	Single Domain
	Turbulence Model	Laminar
	Heat Transfer	Thermal Energy
	Buoyant Flow	
	Boundary Conditions	Symmetry Plane
		Outlet (Subsonic)
		Wall: No-Slip
		Wall: Adiabatic
		Wall: Fixed Temperature
	Output Control	
	Timestep	Transient
	Transient Results File	
CFD-Post	Plots	Default Locators
	Chart	
	Report	
	Other	Time Step Selection

Component	Feature	Details
		Transient Animation

8.2. Overview of the Problem to Solve

The goal of this tutorial is to model a buoyancy-driven flow which require the inclusion of gravitational effects.

The model is a 2D partitioned cavity containing air with properties defined at 25°C. The bottom of the cavity is kept at a constant temperature of 75°C, while the top is held constant at 5°C. The cavity is also tilted at an angle of 30 degrees to the horizontal.



The overall approach for solving this problem is to set up a transient simulation to see how the flow develops starting from stationary conditions. Since you are starting from stationary conditions, there is no need to solve a steady-state simulation for use as the initial guess. You will then model the buoyant flow and create a report outlining the results in CFD-Post. You will also create an animation to see changes in temperature with time.

The provided mesh was created in CFX-4.

8.3. Before You Begin

If this is the first tutorial you are working with, it is important to review the following topics before beginning:

- *Preparing a Working Directory and Starting ANSYS CFX in Standalone Mode* (p. 1)
- *Running ANSYS CFX Tutorials in ANSYS Workbench* (p. 2)
- *Changing the Display Colors* (p. 5)
- *Playing a Tutorial Session File* (p. 4)

8.4. Starting CFX-Pre

1. Prepare the working directory using the following files in the `examples` directory:
   - `Buoyancy2D.geo`
   - `Buoyancy2D.pre`
2. Set the working directory and start CFX-Pre.

   For details, see *Preparing a Working Directory and Starting ANSYS CFX in Standalone Mode* (p. 1)

8.5. Defining a Case in CFX-Pre

If you want to set up the simulation automatically using a tutorial session file, run `Buoyancy2D.pre`. For details, see *Playing a Tutorial Session File* (p. 4). Then proceed to *Obtaining a Solution Using CFX-Solver Manager* (p. 139).

If you want to set up the simulation manually, you are going to import a hexahedral mesh originally generated in CFX-4. The mesh contains labelled regions which will enable you to apply the relevant boundary conditions for this problem.

1. In CFX-Pre, select **File** > **New Case**.
2. Create a new case using **General Mode**.
3. Select **File** > **Save Case As** and set **File name** to `Buoyancy2D`.
4. Click **Save**.

8.5.1. Importing the Mesh

1. Edit `Case Options` > `General` in the **Outline** tree view and ensure that **Automatic Default Domain** is turned off.

   Default Domain generation should be turned off because you will create a new domain manually, later in this tutorial.
2. Right-click `Mesh` and select **Import Mesh** > **Other**. The **Import Mesh** dialog box appears.
3. Apply the following settings:

Setting	Value
File name	Buoyancy2D.geo [1] (p. 133)
Files of type	CFX-4 (*geo)
Options > Mesh Units	m

Footnote

1. This file is in your tutorial directory.

4. Click **Open**.

8.5.2. Analysis Type

The default units and coordinate frame settings are suitable for this tutorial, but the analysis type needs to be set to transient.

You will notice physics validation messages as the case is set to `Transient`. These errors will be fixed later in the tutorial.

1. Click *Analysis Type* .
2. Apply the following settings:

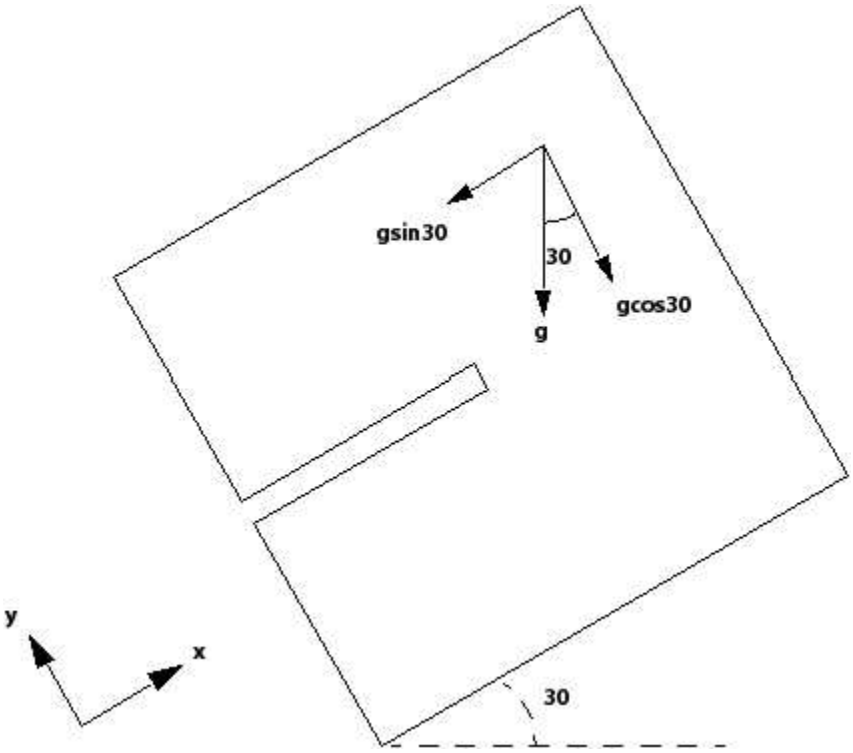
Tab	Setting	Value
Basic Settings	Analysis Type > Option	Transient
	Analysis Type > Time Duration > Total Time	2 [s] [1 (p. 134)]
	Analysis Type > Time Steps > Timesteps	0.025 [s] [2 (p. 134)]
	Analysis Type > Initial Time > Time	0 [s]

Footnotes

1. The total time is the total duration, in real time, for the simulation. For this tutorial, the total time will be set to 2 seconds since it is the time period we are interested in.
2. In this example the simulation moves forward in 0.025 s increments until the total time is reached. The step size was determined as a function of the temperature difference $(T_{\max}-T_{\min})$ between the top and bottom of the cavity, and the length scale of the model (L_{scale}), according to: $\Delta t_g=\sqrt{L_{\text{scale}}/(|\mathbf{g}|\,\beta(T_{\max}-T_{\min}))}$, where $\mathbf{g}$ is the gravity vector and β is the thermal expansivity. For details on computing a fluid time scale estimate, see the theory guide.

3. Click **OK**.

8.5.3. Creating the Domain



You will model the cavity as if it were tilted at an angle of 30°. You can do this by specifying horizontal and vertical components of the gravity vector, which are aligned with the default coordinate axes, as shown in the diagram above.

8.5.3.1. To Create a New Domain

1. Ensure that `Flow Analysis 1` > `Default Domain` is deleted. If not, right-click `Default Domain` and select **Delete**.
2. Click *Domain*  , and set the name to `Buoyancy2D`.
3. Apply the following settings to Buoyancy2D

Tab	Setting	Value
Basic Settings	Location and Type > Location	Primitive 3D
	Fluid and Particle Definitions	Fluid 1
	Fluid and Particle Definitions > Fluid 1 > Material	Air at 25 C
	Domain Models > Pressure > Reference Pressure	1 [atm]
	Domain Models > Buoyancy > Option	Buoyant
	Domain Models > Buoyancy > Gravity X Dirn.	-4.9 [m s^-2]
	Domain Models > Buoyancy > Gravity Y Dirn.	-8.5 [m s^-2]
	Domain Models > Buoyancy > Gravity Z Dirn.	0.0 [m s^-2] [1 (p. 136)]
	Domain Models > Buoyancy > Buoy. Ref. Temp.	40 [C] [2 (p. 136)]

Tab	Setting	Value
Fluid Models	Heat Transfer > Option	Thermal Energy
	Turbulence > Option	None (Laminar)

Footnotes

1. This produces a gravity vector that simulates the tilt of the cavity.
2. Ensure that the unit setting is correct. This is just an approximate representative domain temperature.

Initialization will be set up using *Global Initialization*  , so there is no need to visit the **Initialization** tab.

4. Click **OK**.

8.5.4. Creating the Boundaries

8.5.4.1. Hot and Cold Wall Boundary

Create a wall boundary with a fixed temperature of 75 °C on the bottom surface of the cavity, as follows:

1. Create a new boundary named `hot`.
2. Apply the following settings

Tab	Setting	Value
Basic Settings	Boundary Type	Wall
	Location	WALLHOT
Boundary Details	Heat Transfer > Option	Temperature
	Heat Transfer > Fixed Temperature	75 [C]

3. Click **OK**.
4. Create a new boundary named `cold`.
5. Apply the following settings

Tab	Setting	Value
Basic Settings	Boundary Type	Wall
	Location	WALLCOLD
Boundary Details	Heat Transfer > Option	Temperature
	Heat Transfer > Fixed Temperature	5 [C]

6. Click **OK**.

8.5.4.2. Symmetry Plane Boundary

A single symmetry plane boundary can be used for the front and back of the cavity. Symmetry, which can make a 3D problem into a 2D problem, can be used when the geometry and mesh are invariant normal to the symmetry surface.

1. Create a new boundary named SymP.
2. Apply the following settings

Tab	Setting	Value
Basic Settings	Boundary Type	Symmetry
	Location	SYMMET1, SYMMET2 [1 (p. 137)]

Footnote

1. Use the **Ctrl** key to select more than one region.

3. Click **OK**.

   The default adiabatic wall boundary will be applied automatically to the remaining boundaries.

8.5.5. Setting Initial Values

You should set initial settings using the `Automatic with Value` option when defining a transient simulation. Using this option, the first run will use the specified initial conditions (the air is at rest with a temperature of 5 °C) while subsequent runs will use results file data for initial conditions.

1. Click *Global Initialization*  .
2. Apply the following settings

Tab	Setting	Value
Global Settings	Initial Conditions > Cartesian Velocity Components > Option	Automatic with Value
	Initial Conditions > Cartesian Velocity Components > U	0 [m s^-1]
	Initial Conditions > Cartesian Velocity Components > V	0 [m s^-1]
	Initial Conditions > Cartesian Velocity Components > W	0 [m s^-1]
	Initial Conditions > Static Pressure > Relative Pressure	0 [Pa]
	Initial Conditions > Temperature > Temperature	5 [C]

3. Click **OK**.

8.5.6. Setting Output Control

1. Click *Output Control*  .
2. Click the **Trn Results** tab.
3. In the **Transient Results** tree view, click *Add new item*  , set **Name** to `Transient Results 1`, and click **OK**.
4. Apply the following settings to `Transient Results 1`:

Setting	Value
Option	Selected Variables
Output Variables List [1 (p. 138)]	Pressure, Temperature, Velocity
Output Frequency > Option	Time Interval
Output Frequency > Time Interval	0.1 [s] [2 (p. 138)]

Footnotes

1. Click the *Ellipsis icon*  to select items if they do not appear in the drop-down list. Use the **Ctrl** key to select multiple items.
2. The **Time Interval** option specifies the simulation time interval between the writing of each file. The time interval will be set to 0.1 s, which is 4 times the time step that was set up earlier. There is no need to set a smaller time interval because it does not affect the solution accuracy. Choosing a smaller time interval would simply result in more output files.

5. Click **OK**.

8.5.7. Setting Solver Control

1. Click *Solver Control*  .
2. Apply the following settings

Tab	Setting	Value
Basic Settings	Advection Scheme > Option	High Resolution
	Convergence Control > Max. Coeff. Loops	5 [1 (p. 139)]
	Convergence Criteria > Residual Type	RMS
	Convergence Criteria > Residual Target	1.E-4 [2 (p. 139)]

Footnotes

1. The maximum coefficient loops option determines the maximum number of iterations per time step. It is recommended to set the maximum coefficient loops to between 3 and 5. For this tutorial, it was chosen to be 5, which ensures no net imbalance.
2. An RMS value of at least 1.E-5 is usually required for adequate convergence, but the default value of 1.E-4 is sufficient for demonstration purposes.

3. Click **OK**.

8.5.8. Writing the CFX-Solver Input (.def) File

1. Click *Define Run*  .
2. Apply the following settings:

Setting	Value
File name	Buoyancy2D.def

3. Click **Save**.

   CFX-Solver Manager automatically starts and, on the **Define Run** dialog box, the **Solver Input File** is set.
4. If using Standalone Mode, quit CFX-Pre, saving the simulation (`.cfx`) file at your discretion.

8.6. Obtaining a Solution Using CFX-Solver Manager

When CFX-Pre has shut down and CFX-Solver Manager has started, you can obtain a solution to the CFD problem by using the following procedure.

Note

Recall that the output displayed on the **Out File** tab of the CFX-Solver Manager is more complicated for transient problems than for steady-state problems. Each timestep consists of several iterations, and after the timestep, information about various quantities is printed.

1. Click **Start Run**.
2. Select the check box next to **Post-Process Results** when the completion message appears at the end of the run.
3. If using Standalone Mode, select the check box next to **Shut down CFX-Solver Manager**.
4. Click **OK**.

8.7. Viewing the Results in CFD-Post

In this section, you will create a customized report in CFD-Post. You will also, optionally, make an animation to see changes in temperature with time.

8.7.1. Simple Report

First, you will view a report that is created with little effort:

1. Click the **Report Viewer** tab. Note that the report loads with some automatically-generated statistical information.
2. In the **Outline** tree view, under `Report`, experiment with the various settings for `Mesh Report`, `Physics Report` and other report objects. These settings control the report contents. On the **Report Viewer** tab, you can click **Refresh** to see the changes to your report.

8.7.2. Plots for Customized Reports

Here, you will create the following objects in preparation for generating a more customized report:

- Contour plot of temperature
- Point locators (for observing temperature)
- Comment
- Figure showing the contour plot and point locator
- Time chart showing the temperature at the point locator
- Table.

8.7.2.1. Contour Plot of Temperature

1. Click the **3D Viewer** tab and right-click a blank area of the viewer, then select **Predefined Camera** > **View From -Z**.
2. Select **Insert** > **Contour** from the main menu.
3. Accept the default name by clicking **OK**.
4. Set **Locations** to SymP.
5. Set **Variable** to `Temperature`.
6. Click **Apply**.

The contour plot shows the temperature at the end of the simulation, since CFD-Post loads values for the last timestep by default. You can load different timesteps using the **Timestep Selector** dialog box, accessible by selecting **Tools** > **Timestep Selector** from the main menu.

Before proceeding, turn off the visibility of the contour plot.

8.7.2.2. Point Locators

Two points will be created to generate a time chart of temperature vs. time later on in this tutorial. The two points were chosen to be located half way in between the bottom and top cavity, close to where the average temperature is going to be.

1. From the main menu, select **Insert** > **Location** > **Point**.
2. Accept the default name by clicking **OK**.
3. Set **Method** to XYZ.
4. Set **Point** coordinates to `0.098`, `0.05`, `0.00125`.

5. Click **Apply**.

   Note the location of `Point 1` in the viewer.
6. Right-click the `Point 1` object in the tree view and select **Duplicate** from the shortcut menu.
7. Accept the default name by clicking **OK**.
8. Right-click the `Point 2` object in the tree view and select **Edit** from the shortcut menu.
9. Change the x-coordinate to `0.052`.
10. Click **Apply**.

    Note the location of `Point 2` in the viewer.

8.7.2.3. Comment

1. Click *Create comment*  .
2. Accept the default name by clicking **OK**.

   A comment object appears in the tree view, under the `Report` object.
3. Set **Heading** to `Buoyant Flow in a Partitioned Cavity`.
4. In the large text box, type:

   ```
   This is a sample paragraph.
   ```

8.7.2.4. Figure

Figures are CCL objects that can be used to store and switch between different views in a given viewport. By selecting a figure, the information contained in the figure, such as the camera angle, zoom level, lighting and the visibility setting of each object in the tree view, is applied to the active viewport and is usable in reports.

1. Click the **3D Viewer** tab.
2. Select **Insert** > **Figure** from the main menu.
3. Accept the default name by clicking **OK**.

   The **Make copies of objects** check box determines whether or not the objects that are visible in the viewer are copied. If objects are copied, then the copies are used in the figure instead of the originals. Since you are not using multiple views or figures, the check box setting does not matter.

   A figure object will appear under the `Report` branch in the tree view.

8.7.2.5. Time Chart of Temperature

Time charts use expressions or a point locator to plot the variation of a scalar value with time. In this tutorial, the variation of temperature versus time will be plotted.

1. Select **Insert** > **Chart** from the main menu.
2. Accept the default name by clicking **OK**.
3. Set **Type** to `XY-Transient or Sequence`.
4. Set **Title** to `Temperature versus Time`.

5. Click the **Data Series** tab.
6. Set **Name** to `Temperature at Point 1`.
7. Set **Location** to `Point 1`.
8. Click the **X Axis** tab.
9. Set **Data Selection** > **Expression** to `Time`.
10. Click the **Y Axis** tab.
11. Set **Data Selection** > **Variable** to `Temperature`.
12. Ensure that the **Refresh chart on Apply** check box (at the bottom of the details view) is selected.

    This causes the chart to be regenerated in the **Chart Viewer** tab each time you apply changes in the details view. (When the **Refresh chart on Apply** check box is cleared, the chart will be regenerated only when you manually refresh it. One way to refresh the chart is to click the **Refresh** button at the top of the **Chart Viewer** tab.)
13. Click **Apply**.

    A chart object will appear under the `Report` branch in the tree view.

    It may take some time for the chart to appear because every transient results file will be loaded in order to generate the time chart.
14. Click **New**  (on the **Data Series** tab).
15. Set **Name** to `Temperature at Point 2`.
16. Set **Location** to `Point 2`.
17. Click **Apply**.

    A second chart line will appear in the chart, representing the temperature at `Point 2`.

8.7.2.6. Table of Temperature Values

1. Select **Insert** > **Table** from the main menu.
2. Accept the default name by clicking **OK**.

   A table object will appear under the `Report` branch in the tree view.
3. Set the following:

Cell	Value
A1	Location
A2	Point 1
A3	Point 2
B1	Temperature
B2	=probe(Temperature)@Point 1
B3	=probe(Temperature)@Point 2

The table shows temperatures at the end of the simulation, since CFD-Post loads values for the last timestep by default. You can load different timesteps using the **Timestep Selector** dialog box, accessible by selecting **Tools** > **Timestep Selector**.

8.7.3. Customized Report

Right-click the `Report` object and select **Refresh Preview** from the shortcut menu. Look at the report in the **Report Viewer** tab. Note that, in addition to the automatically-generated objects that you saw earlier when creating a simple report, this report also includes the customized figure, time chart and table described above.

8.7.4. Animations

You may want to create an animation of the buoyant flow over time. Use the animation feature to see the changing temperature field. The animation feature was used in *Flow from a Circular Vent* (p. 93).

8.7.5. Completion

When you have finished, quit CFD-Post.

Chapter 9: Free Surface Flow Over a Bump

This tutorial includes:

9.1. Tutorial Features

In this tutorial you will set up a 2D problem in which you:

- Import a mesh.
- Set up appropriate boundary conditions for a free surface simulation. (Free surface simulations are more sensitive to incorrect boundary and initial guess settings than other more basic models.)
- Use mesh adaption to refine the mesh where the volume fraction gradient is greatest. (The refined mesh aids in the development of a sharp interface between the liquid and gas.)

Component	Feature	Details
CFX-Pre	User Mode	General Mode
	Analysis Type	Steady State
	Fluid Type	General Fluid
	Domain Type	Single Domain
	Turbulence Model	k-Epsilon
	Heat Transfer	Isothermal
	Buoyant Flow	
	Multiphase	Homogeneous Model
	Boundary Conditions	Inlet
		Opening
		Outlet
		Symmetry Plane
		Wall: No Slip
	CEL (CFX Expression Language)	
	Mesh Adaption	
	Timestep	Physical Time Scale

Component	Feature	Details
CFD-Post	Plots	Default Locators
		Isosurface
		Polyline
		Sampling Plane
		Vector
		Volume
	Other	Chart Creation
		Title/Text
		Viewing the Mesh

9.2. Overview of the Problem to Solve

This tutorial demonstrates the simulation of a free surface flow.

The geometry consists of a 2D channel in which the bottom of the channel is interrupted by a semicircular bump of radius 30 mm. The problem environment comprises air at 1 Pa and isothermal water; the normal inlet speed is 0.26 m/s; the incoming water has a turbulence intensity of 5%. The flow upstream of the bump is subcritical. The downstream boundary conditions (the height of the water) were estimated for this tutorial; you can do this using an analytical 1D calculation or data tables for flow over a bump.

9.3. Approach to the Problem

A mesh is provided. You will create a two-phase homogeneous setting and the expressions that will be used in setting initial values and boundary conditions. Later, you will use mesh adaption to improve the accuracy of the downstream simulation.

9.4. Before You Begin

If this is the first tutorial you are working with, it is important to review the following topics before beginning:

- *Preparing a Working Directory and Starting ANSYS CFX in Standalone Mode* (p. 1)
- *Running ANSYS CFX Tutorials in ANSYS Workbench* (p. 2)
- *Changing the Display Colors* (p. 5)
- *Playing a Tutorial Session File* (p. 4)

9.5. Starting CFX-Pre

1. Prepare the working directory using the following files in the `examples` directory:

   - `Bump2D.pre`
   - `Bump2DExpressions.ccl`
   - `Bump2Dpatran.out`

2. Set the working directory and start CFX-Pre.

   For details, see *Preparing a Working Directory and Starting ANSYS CFX in Standalone Mode* (p. 1)

9.6. Defining a Simulation in CFX-Pre

If you want to set up the simulation automatically using a tutorial session file, run `Bump2D.pre`. For details, see *Playing a Tutorial Session File* (p. 4). Then proceed to *Obtaining a Solution Using CFX-Solver Manager* (p. 155).

If you want to set up the simulation manually, proceed to the following steps:

1. In CFX-Pre, select **File** > **New Case**.
2. Select **General** and click **OK**.
3. Select **File** > **Save Case As**.
4. Under **File name**, type `Bump2D`.
5. Click **Save**.

9.6.1. Importing the Mesh

1. Right-click `Mesh` and select **Import Mesh** > **Other**. The **Import Mesh** dialog box appears.
2. Apply the following settings:

Setting	Value
File name	Bump2Dpatran.out
Files of type	PATRAN Neutral (*out *neu)
Options > Mesh units	m

3. Click **Open**.
4. To best orient the view, right-click a blank area in the viewer and select **Predefined Camera** > **View From -Z** from the shortcut menu.

9.6.1.1. Viewing the Region Labels

Enable the display of region labels so that you can see where you will define boundaries later in this tutorial:

1. In the **Outline** tree view, edit `Case Options` > `Labels and Markers`.
2. Apply the following settings:

Tab	Setting	Value
Settings	Show Labels	(Selected)
	Show Labels > Show Primitive 3D Labels	(Selected)
	Show Labels > Show Primitive 2D Labels	(Selected)

3. Click **OK**.

9.6.2. Creating Expressions for Initial and Boundary Conditions

Simulation of free surface flows usually requires defining boundary and initial conditions to set up appropriate pressure and volume fraction fields. You will need to create expressions using CEL (CFX Expression Language) to define these conditions.

In this simulation, the following conditions are set and require expressions:

- An inlet boundary where the volume fraction above the free surface is 1 for air and 0 for water, and below the free surface is 0 for air and 1 for water.
- A pressure-specified outlet boundary, where the pressure above the free surface is constant and the pressure below the free surface is a hydrostatic distribution. This requires you to know the approximate height of the fluid at the outlet. In this case, an analytical solution for 1D flow over a bump was used to determine the value for `DownH` in *Creating Expressions in CEL* (p. 148). The simulation is not sensitive to the exact outlet fluid height, so an approximation is sufficient. You will examine the effect of the outlet boundary condition in the postprocessing section and confirm that it does not affect the validity of the results. It is necessary to specify such a boundary condition to force the flow downstream of the bump into the supercritical regime.
- An initial pressure field for the domain with a similar pressure distribution to that of the outlet boundary.

Either create expressions using the **Expressions** workspace or read in expressions from the example file provided:

- *Creating Expressions in CEL* (p. 148)
- *Reading Expressions From a File* (p. 149)

9.6.2.1. Creating Expressions in CEL

The expressions you create in this step are the same as the ones provided in *Reading Expressions From a File* (p. 149), so you can choose to follow either set of instructions.

1. Right-click `Expressions, Functions and Variables` > `Expressions` in the tree view and select **Insert** > **Expression**.
2. Set the name to `UpH` and click **OK** to create the upstream free surface height.
3. Set **Definition** to `0.069 [m]`, and then click **Apply**.
4. Use the same method to create the expressions listed in the table below. These are expressions for the downstream free surface height, the fluid density, the buoyancy reference density, the calculated density of the fluid (density - buoyancy reference density), the upstream volume fractions of air and water, the upstream pressure distribution, the downstream volume fractions of air and water, and the downstream pressure distribution.

Name	Definition
DownH	0.022 [m]
DenWater	997 [kg m^-3]
DenRef	1.185 [kg m^-3]
DenH	(DenWater - DenRef)
UpVFAir	step((y-UpH)/1[m])
UpVFWater	1-UpVFAir
UpPres	DenH*g*UpVFWater*(UpH-y)

Name	Definition
DownVFAir	step((y-DownH)/1[m])
DownVFWater	1-DownVFAir
DownPres	DenH*g*DownVFWater*(DownH-y)

5. Proceed to *Creating the Domain* (p. 149).

9.6.2.2. Reading Expressions From a File

1. If you have not done so already, copy the file `<CFXROOT>/examples/Bump2DExpressions.ccl` to your working directory.
2. Select **File** > **Import** > **CCL**.
3. In the **Import CCL** dialog box, ensure that the **Append** option is selected.
4. Select `Bump2DExpressions.ccl`.
5. Click **Open**.
6. After the file has been imported, use the **Expressions** tree view to view the expressions that have been created.

9.6.3. Creating the Domain

Set up a homogeneous, two-fluid environment:

1. Edit `Case Options` > `General` in the **Outline** tree view and ensure that **Automatic Default Domain** is turned on. A domain named `Default Domain` should appear under the `Simulation` > `Flow Analysis 1` branch.
2. Double-click **Default Domain**.
3. Under **Fluid and Particle Definitions**, delete `Fluid 1` and create a new fluid named `Air`.
4. Confirm that the following settings are configured:

Tab	Setting	Value
Basic Settings	Fluid and Particle Definitions	Air
	Fluid and Particle Definitions > Air > Material	Air at 25 C

5. Click *Add new item*  and create a new fluid named `Water`.
6. Apply the following settings:

Tab	Setting	Value
Basic Settings	Fluid and Particle Definitions	Water
	Fluid and Particle Definitions > Water > Material	Water [1 (p. 150)]
	Domain Models > Pressure > Reference Pressure	1 [atm]
	Domain Models > Buoyancy > Option	Buoyant
	Domain Models > Buoyancy > Gravity X Dirn.	0 [m s^-2]

Tab	Setting	Value
	Domain Models > Buoyancy > Gravity Y Dirn.[2 (p. 150)]	-g
	Domain Models > Buoyancy > Gravity Z Dirn.	0 [m s^-2]
	Domain Models > Buoyancy > Buoy. Ref. Density [3 (p. 150)]	DenRef
Fluid Models	Multiphase > Homogeneous Model [4 (p. 150)]	(Selected)
	Multiphase > Free Surface Model > Option	Standard
	Heat Transfer > Option	Isothermal
	Heat Transfer > Fluid Temperature	25 [C]
	Turbulence > Option	k-Epsilon

Footnotes

1. The models selected here describe how the fluids interact. No mass transfer between the phases occurs in this example. You do not need to model surface tension.
2. You need to click *Enter Expression*  beside the field first.
3. Always set Buoyancy Reference Density to the density of the least dense fluid in free surface calculations.
4. The homogeneous model solves for a single solution field.

7. Click **OK**.

9.6.4. Creating the Boundaries

9.6.4.1. Inlet Boundary

1. Create a new boundary named `inflow`.
2. Apply the following settings (from the problem description):

Tab	Setting	Value
Basic Settings	Boundary Type	Inlet
	Location	INFLOW
Boundary Details	Mass and Momentum > Option	Normal Speed
	Mass and Momentum > Normal Speed	0.26 [m s^-1]
	Turbulence > Option	Intensity and Length Scale
	Turbulence > Fractional Intensity	0.05
	Turbulence > Eddy Length Scale [1 (p. 151)]	UpH
Fluid Values	Boundary Conditions	Air
	Boundary Conditions > Air > Volume Fraction > Volume Fraction	UpVFAir

Tab	Setting	Value
	Boundary Conditions	Water
	Boundary Conditions > Water > Volume Fraction > Volume Fraction	UpVFWater

Footnote

1. Click the *Enter Expression* icon  .

3. Click **OK**.

9.6.4.2. Outlet Boundary

1. Create a new boundary named `outflow`.
2. Apply the following settings:

Tab	Setting	Value
Basic Settings	Boundary Type	Outlet
	Location	OUTFLOW
Boundary Details	Flow Regime > Option	Subsonic
	Mass and Momentum > Option	Static Pressure
	Mass and Momentum > Relative Pressure	DownPres

3. Click **OK**.

9.6.4.3. Symmetry Boundaries

1. Create a new boundary named `front`.
2. Apply the following settings:

Tab	Setting	Value
Basic Settings	Boundary Type	Symmetry [1 (p. 151)]
	Location	FRONT

Footnote

1. Symmetry, which makes a 3D problem into a 2D problem, can be used when geometry and mesh are invariant normal to the symmetry surface.

3. Click **OK**.
4. Create a new boundary named `back`.
5. Apply the following settings:

Tab	Setting	Value
Basic Settings	Boundary Type	Symmetry
	Location	BACK

6. Click **OK**.

9.6.4.4. Opening and Wall Boundaries

1. Create a new boundary named `top`.
2. Apply the following settings:

Tab	Setting	Value
Basic Settings	Boundary Type	Opening
	Location	TOP
Boundary Details	Mass And Momentum > Option	Entrainment
	Mass And Momentum > Relative Pressure	0 [Pa]
	Turbulence > Option	Zero Gradient
Fluid Values	Boundary Conditions	Air
	Boundary Conditions > Air > Volume Fraction > Volume Fraction	1.0
	Boundary Conditions	Water
	Boundary Conditions > Water > Volume Fraction > Volume Fraction	0.0

3. Click **OK**.
4. Create a new boundary named `bottom`.
5. Apply the following settings:

Tab	Setting	Value
Basic Settings	Boundary Type	Wall
	Location	BOTTOM1, BOTTOM2, BOTTOM3
Boundary Details	Mass and Momentum > Option	No Slip Wall
	Wall Roughness > Option	Smooth Wall

6. Click **OK**.

9.6.5. Setting Initial Values

Set up the initial values to be consistent with the inlet boundary conditions:

1. Click *Global Initialization* .

2. Apply the following settings:[1]

Tab	Setting	Value
Global Settings	Initial Conditions > Cartesian Velocity Components > Option	Automatic with Value
	Initial Conditions > Cartesian Velocity Components > U	0.26 [m s^-1]
	Initial Conditions > Cartesian Velocity Components > V	0 [m s^-1]
	Initial Conditions > Cartesian Velocity Components > W	0 [m s^-1]
	Initial Conditions > Static Pressure > Option	Automatic with Value
	Initial Conditions > Static Pressure > Relative Pressure	UpPres
Fluid Settings	Fluid Specific Initialization	Air
	Fluid Specific Initialization > Air > Initial Conditions > Volume Fraction > Option	Automatic with Value
	Fluid Specific Initialization > Air > Initial Conditions > Volume Fraction > Volume Fraction	UpVFAir
	Fluid Specific Initialization	Water
	Fluid Specific Initialization > Water > Initial Conditions > Volume Fraction > Option	Automatic with Value
	Fluid Specific Initialization > Water > Initial Conditions > Volume Fraction > Volume Fraction	UpVFWater

3. Click **OK**.

9.6.6. Setting Mesh Adaption Parameters

To improve the resolution of the interface between the air and the water, set up the mesh adaption settings:

1. Click *Mesh Adaption*  .
2. Apply the following settings:

Tab	Setting	Value
Basic Settings	Activate Adaption	(Selected)
	Save Intermediate Files	(Cleared)
	Adaption Criteria > Variables List	Air.Volume Fraction
	Adaption Criteria > Max. Num. Steps	2

[1]The values are from the problem specification.

Tab	Setting	Value
	Adaption Criteria > Option	Multiple of Initial Mesh
	Adaption Criteria > Node Factor	4
	Adaption Convergence Criteria > Max. Iter. per Step	100
Advanced Options	Node Alloc. Param.	1.6
	Number of Levels	2

Note

You can find descriptions of the mesh adaption process and of the parameters you are setting in Mesh Adaption in the *CFX-Pre User's Guide*. The values used here were determined through experimentation.

3. Click **OK**.

9.6.7. Setting the Solver Controls

Note

Setting **Max. Iterations** to 200 (below) and **Number of (Adaption) Levels** to 2 with a **Max. Iter. per Step** of 100 timesteps each (in the previous section), results in a total maximum number of timesteps of 400 (2*100+200=400).

1. Click *Solver Control*  .
2. Apply the following settings:

Tab	Setting	Value
Basic Settings	Convergence Control > Max. Iterations	200
	Convergence Control >Fluid Timescale Control > Timescale Control	Physical Timescale
	Convergence Control >Fluid Timescale Control > Physical Timescale	0.25 [s] [1 (p. 154)]
Advanced Options	Multiphase Control	(Selected)
	Multiphase Control > Volume Fraction Coupling	(Selected)
	Multiphase Control > Volume Fraction Coupling > Option	Coupled

Footnote

1. This value is based on the time it takes the water to flow over the bump.

Note

Selecting these options on the solver control activates the Coupled Volume Fraction solution algorithm. This algorithm typically converges better than the Segregated Volume Faction algorithm for buoyancy-driven problems such as this tutorial. The Segregated Volume Faction algorithm would have required a 0.05 second timescale, as compared with 0.25 seconds for the Coupled Volume Fraction algorithm.

3. Click **OK**.

9.6.8. Writing the CFX-Solver Input (.def) File

1. Click *Define Run*  .
2. Apply the following settings:

Setting	Value
File name	`Bump2D.def`

3. Click **Save**.

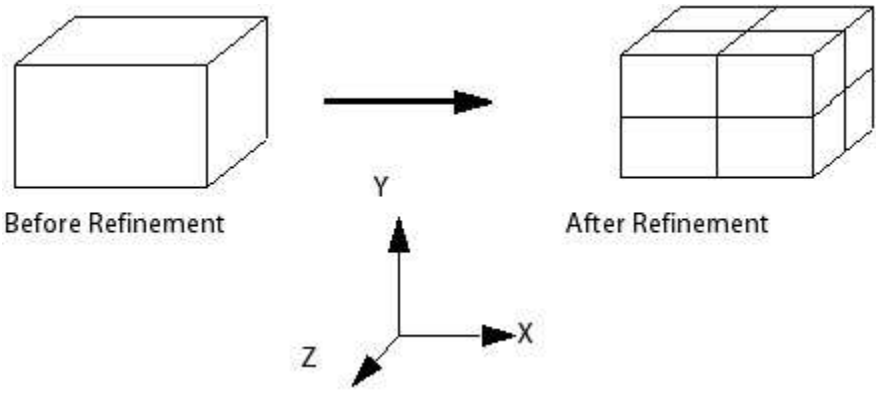
   CFX-Solver Manager automatically starts and, on the **Define Run** dialog box, the **Solver Input File** is set.
4. If using Standalone Mode, quit CFX-Pre, saving the simulation (`.cfx`) file at your discretion.

9.7. Obtaining a Solution Using CFX-Solver Manager

Click **Start Run**.

Within 100 iterations after CFX-Solver Manager has started, the first adaption step is performed. Information written to the `.out` file includes the number of elements refined and the size of the new mesh.

After mesh refinement, there is a jump in the residual levels. This is because the solution from the old mesh is interpolated onto the new mesh. A new residual plot also appears for the W-Mom-Bulk equation. Hexahedral mesh elements are refined orthogonally, so the mesh is no longer 2D (it is more than 1 element thick in the z-direction). Convergence to the target residual level is achieved.



It is common for convergence in a residual sense to be difficult to obtain in a free surface simulation, due to the presence of small waves at the surface preventing the residuals from dropping to the target level. This is more frequently a problem in the subcritical flow regime, as the waves can travel upstream. In the supercritical regime, the waves tend to get carried downstream and out the domain. To satisfy convergence in these cases, monitor the value of a global quantity (for example, drag for flow around a ship's hull) to see when a steady state value is reached.

Where there is no obvious global quantity to monitor, you should view the results to see where the solution is changing. You can do this by running transient (with timesteps that are small enough to capture transient effects) for a few timesteps, starting from a results file that you think is converged or from backup results files you have written at different timesteps.

In both cases, look to see where the results are changing (this could be due to the presence of small transient waves). Also confirm that the value of quantities that you are interested in (for example, downstream fluid height for this case) has reached a steady-state value.

1. When a dialog box appears at the end of the run, select **Post-Process Results**.
2. If using Standalone Mode, select **Shut down CFX-Solver Manager**.
3. Click **OK**.

9.8. Viewing the Results in CFD-Post

Display the distribution of volume fraction of water in the domain:

1. To best orient the view, right-click on a blank area in the viewer and select **Predefined Camera** > **View From -Z**.
2. Zoom in so the geometry fills the viewer.
3. In the tree view, edit `Bump2D_001` > `Default Domain` > `front`.
4. Apply the following settings:

Tab	Setting	Value
Color	Mode	Variable
	Variable	Water.Volume Fraction

5. Click **Apply**.
6. Clear the check box next to `front`.

9.8.1. Creating Velocity Vector Plots

The next step involves creating a sampling plane upon which to display velocity vectors for `Water`.

1. Select **Insert** > **Location** > **Plane** to create a new plane named `Plane 1`.
2. Apply the following settings:

Tab	Setting	Value
Geometry	Definition > Method	XY Plane
	Plane Bounds > Type	Rectangular
	Plane Bounds > X Size	1.25 [m] [1 (p. 157)]

Tab	Setting	Value
	Plane Bounds > Y Size	0.3 [m]
	Plane Bounds > X Angle	0 [degree]
	Plane Type	Sample
	X Samples	160 [2 (p. 157)]
	Y Samples	40
Render	Show Faces	(Cleared)
	Show Mesh Lines	(Selected)

Footnotes

1. The Plane Bounds settings overlap the plane with the wireframe. You can experiment with other values and click **Apply** to see the results.
2. The Plane Samples settings produce square elements. You can experiment with other values and click **Apply** to see the results.

3. Click **Apply**.
4. Clear the check box next to `Plane 1`.
5. Create a new vector named `Vector 1`.
6. Apply the following settings:

Tab	Setting	Value
Geometry	Definition > Locations	Plane 1
	Definition > Variable [1 (p. 157)]	Water.Velocity
Symbol	Symbol Size	0.5

Footnote

1. Because fluids in a free-surface calculation share the same velocity field, only the velocity of the first non-vapor fluid is available. The other allowed velocities are superficial velocities. For details, see *Further Postprocessing* (p. 162).

7. Click **Apply**.
8. Apply the following settings:

Tab	Setting	Value
Geometry	Definition > Variable	Air.Superficial Velocity
Symbol	Symbol Size	0.15
	Normalize Symbols	(Selected)

9. Click **Apply**.

9.8.2. Viewing Mesh Refinement

In this section, you will view the surface mesh on one of the symmetry boundaries, create volume objects to show where the mesh was modified, and create a vector plot to visualize the added mesh nodes.

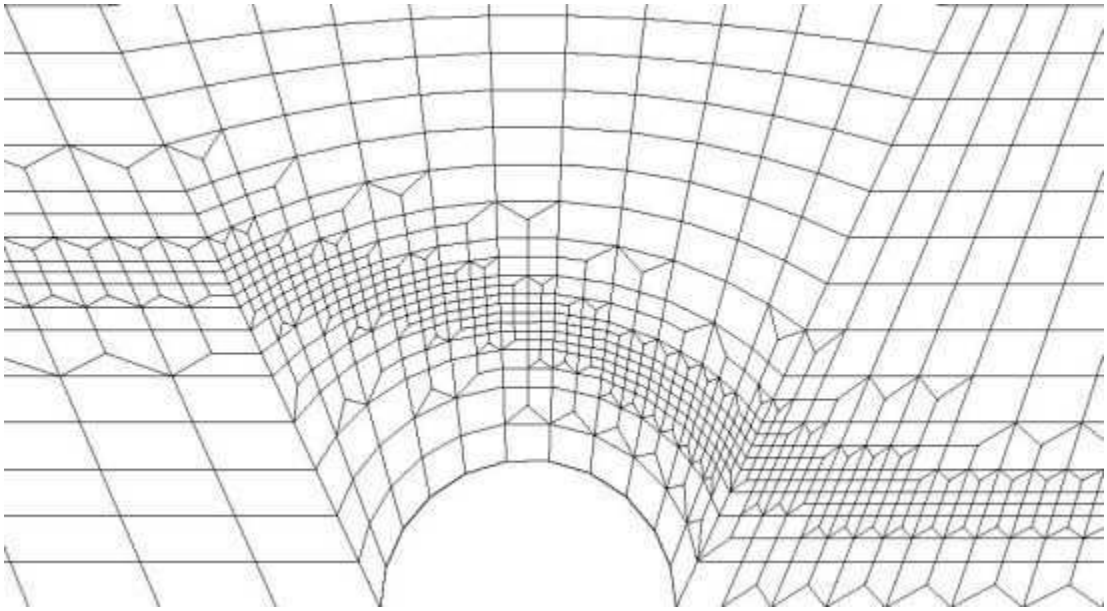
1. Clear the check box next to `Vector 1`.
2. Zoom in so the geometry fills the Viewer.
3. In **Outline** under `Default Domain`, edit `front`.
4. Apply the following settings:

Tab	Setting	Value
Color	Mode	Constant
Render	Show Faces	(Cleared)
	Show Mesh Lines	(Selected)

5. Click **Apply**.

   - The mesh has been refined near the free surface.
   - In the transition region between different levels of refinement, tetrahedral and pyramidal elements are used because it is not possible to recreate hexahedral elements in CFX. Near the inlet, the aspect ratio of these elements increases.
   - Avoid performing mesh refinement on high-aspect-ratio hex meshes as this will produce high aspect ratio tetrahedral-elements and result in poor mesh quality.

Figure 9.1 Mesh around the bump



6. Create a new volume named `first refinement elements`.
7. Apply the following settings:

Tab	Setting	Value
Geometry	Definition > Method	Isovolume
	Definition > Variable	Refinement Level [1 (p. 159)]
	Definition > Mode	At Value
	Definition > Value	1
Render	Show Faces	(Cleared)
	Show Mesh Lines	(Selected)
	Show Mesh Lines > Line Width	2
	Show Mesh Lines > Color Mode	User Specified
	Show Mesh Lines > Line Color	(Green)

Footnote

1. Click *More variables* ... to access the Refinement Level value.

8. Click **Apply**.

   You will see a band of green, which indicates the elements that include nodes added during the first mesh adaption.

9. Create a new volume named `second refinement elements`.

10. Apply the following settings:

Tab	Setting	Value
Geometry	Definition > Method	Isovolume
	Definition > Variable	Refinement Level
	Definition >Mode	At Value
	Definition > Value	2
Color	Color	White
Render	Show Faces	(Selected)
	Show Mesh Lines	(Selected)
	Show Mesh Lines > Line Width	4
	Show Mesh Lines > Color Mode	User Specified
	Show Mesh Lines > Line Color	(Black)

11. Click **Apply**.

    You will see a band of white (with black lines); this indicates the elements that include nodes added during the second mesh adaption.

12. Zoom in to a region where the mesh has been refined.

The Refinement Level variable holds an integer value at each node, which is either 0, 1, or 2 (because you used a maximum of two adaption levels).

The nodal values of refinement level will be visualized next.

13. Create a new vector named `Vector 2`.
14. Apply the following settings:

Tab	Setting	Value
Geometry	Definition > Locations	Default Domain
	Definition > Variable [1 (p. 160)]	(Any Vector Variable)
Color	Mode	Variable
	Variable	Refinement Level
Symbol	Symbol	Cube
	Symbol Size	0.02
	Normalize Symbols	(Selected)

Footnote

1. The variable's magnitude and direction do not matter because you will change the vector symbol to a cube with a normalized size.

15. Click **Apply**.

In `Vector 2`, Blue nodes (Refinement Level 0 according to the color legend) are part of the original mesh. Green nodes (Refinement Level 1) were added during the first adaption step. Red nodes (Refinement Level 2) were added during the second adaption step. Note that some elements contain combinations of blue, green, and red nodes.

9.8.3. Creating an Isosurface to Show the Free Surface

Later in this tutorial, you will create a chart to show the variation in free surface height along the channel. The data for the chart will be sampled along a polyline that follows the free surface. To make the polyline, you will use the intersection between one of the symmetry planes and an isosurface that follows the free surface. Start by creating an isosurface on the free surface:

1. Turn off the visibility for all objects except `Wireframe`.
2. Create a new isosurface named `Isosurface 1`.
3. Apply the following settings:

Tab	Setting	Value
Geometry	Definition > Variable	Water.Volume Fraction
	Definition > Value	0.5

4. Click **Apply**.

Creating isosurfaces using this method is a good way to visualize a free surface in a 3D simulation.

5. Right-click any blank area in the viewer, select **Predefined Camera**, then select **Isometric View (Y up)**.

9.8.4. Creating a Polyline that Follows the Free Surface

Create a polyline along the isosurface that you created in the previous step:

1. Turn off the visibility of `Isosurface 1`.
2. Create a new polyline named `Polyline 1`.
3. Apply the following settings:

Tab	Setting	Value
Geometry	Method	Boundary Intersection
	Boundary List	front
	Intersect With	Isosurface 1

4. Click **Apply**.

   A green line is displayed that follows the high-Z edge of the isosurface.

9.8.5. Creating a Chart to Show the Height of the Surface

Create a chart that plots the free surface height using the polyline that you created in the previous step:

1. Create a new chart named `Chart 1`.

   The **Chart Viewer** tab is selected.
2. Apply the following settings:

Tab	Setting	Value
General	Title	Free Surface Height for Flow over a Bump
Data Series	Name	free surface height
	Location	Polyline 1
X Axis	Variable	X
Y Axis	Variable	Y
Line Display	Symbols	Rectangle

3. Click **Apply**.

As discussed in *Creating Expressions for Initial and Boundary Conditions* (p. 148), an approximate outlet elevation is imposed as part of the boundary, even though the flow is supercritical. The chart illustrates the effect of this, in that the water level rises just before the exit plane. It is evident from this plot that imposing the elevation does not affect the upstream flow.

The chart shows a wiggle in the elevation of the free surface interface at the inlet. This is related to an overspecification of conditions at the inlet because both the inlet velocity and elevation were specified. For a subcritical inlet, only the velocity or the total energy should be specified. The wiggle is due to a small inconsistency between the specified elevation and the elevation computed by the solver to obtain critical conditions

at the bump. The wiggle is analogous to one found if pressure and velocity were both specified at a subsonic inlet in a converging-diverging nozzle with choked flow at the throat.

9.8.6. Further Postprocessing

You may want to create some plots using the `<Fluid>.Superficial Velocity` variables. This is the fluid volume fraction multiplied by the fluid velocity and is sometimes called the volume flux. It is useful to use this variable for vector plots in separated multiphase flow, as you will only see a vector where a significant amount of that phase exists.

Tip

You can right-click on an existing vector plot and select a new vector variable.

9.9. Using a Supercritical Outlet Condition

For supercritical free surface flows, the supercritical outlet boundary is usually the most appropriate boundary for the outlet because it does not rely on the specification of the outlet pressure distribution (which depends on an estimate of the free surface height at the outlet). The supercritical outlet boundary requires a relative pressure specification for the gas only; no pressure information is required for the liquid at the outlet. For this tutorial, the relative gas pressure at the outlet should be set to 0 Pa.

The supercritical outlet condition may admit multiple solutions. To find the supercritical solution, it is often necessary to start with a static pressure outlet condition (as previously done in this tutorial) or an average static pressure condition where the pressure is set consistent with an elevation to drive the solution into the supercritical regime. The outlet condition can then be changed to the supercritical option.

Chapter 10: Supersonic Flow Over a Wing

This tutorial includes:

10.1. Tutorial Features

In this tutorial you will learn about:

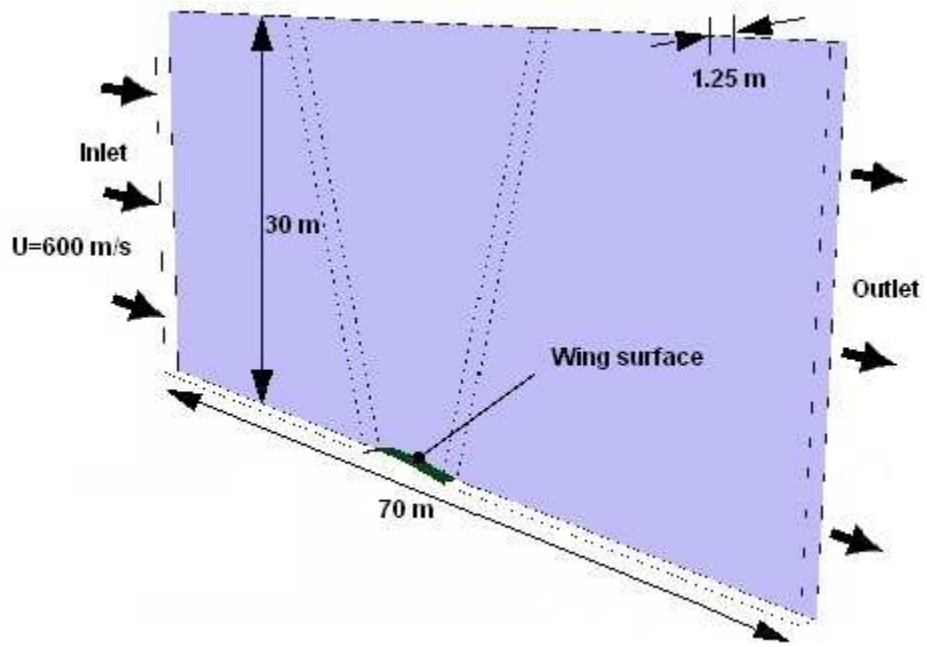
- Setting up a supersonic flow simulation.
- Using the Shear Stress Transport turbulence model to accurately resolve flow around a wing surface.
- Defining a custom vector to display pressure distribution.

Component	Feature	Details
CFX-Pre	User Mode	General Mode
	Analysis Type	Steady State
	Fluid Type	Air Ideal Gas
	Domain Type	Single Domain
	Turbulence Model	Shear Stress Transport
	Heat Transfer	Total Energy
	Boundary Conditions	Inlet (Supersonic)
		Outlet (Supersonic)
		Symmetry Plane
		Wall: No-Slip
		Wall: Adiabatic
		Wall: Free-Slip
	Domain Interfaces	Fluid-Fluid (No Frame Change)
	Timestep	Maximum Timescale
CFD-Post	Plots	Contour
		Vector
	Other	Variable Details View

10.2. Overview of the Problem to Solve

This example demonstrates the use of CFX in simulating supersonic flow over a symmetric NACA0012 airfoil at 0° angle of attack. A 2D section of the wing is modeled. A 2D hexahedral mesh is provided that you will import into CFX-Pre.

The environment is 300 K air at 1 atmosphere that passes the wing at 600 m/s. The turbulence intensity is low (.01) with an eddy length scale of .02 meters.



10.3. Approach to the Problem

A mesh is provided. You will create a domain that contains three regions that will be connected by fluid-fluid interfaces. To solve the simulation, you will start with a conservative time scale that gradually increases towards the fluid residence time as the residuals decrease.

10.4. Before You Begin

If this is the first tutorial you are working with, it is important to review the following topics before beginning:

- *Preparing a Working Directory and Starting ANSYS CFX in Standalone Mode* (p. 1)
- *Running ANSYS CFX Tutorials in ANSYS Workbench* (p. 2)
- *Changing the Display Colors* (p. 5)
- *Playing a Tutorial Session File* (p. 4)

10.5. Starting CFX-Pre

1. Prepare the working directory using the following files in the `examples` directory:
   - `WingSPSMesh.out`
   - `WingSPS.pre`

2. Set the working directory and start CFX-Pre.

   For details, see *Preparing a Working Directory and Starting ANSYS CFX in Standalone Mode* (p. 1)

10.6. Defining a Case in CFX-Pre

If you want to set up the simulation automatically using a tutorial session file, run `WingSPS.pre`. For details, see *Playing a Tutorial Session File* (p. 4). Then proceed to *Obtaining a Solution Using CFX-Solver Manager* (p. 170).

If you want to set up the simulation manually, proceed to the following steps:

1. In CFX-Pre, select **File** > **New Case**.
2. Select **General** and click **OK**.
3. Select **File** > **Save Case As**.
4. Under **File name**, type `WingSPS`.
5. Click **Save**.

10.6.1. Importing the Mesh

1. Right-click `Mesh` and select **Import Mesh** > **Other**. The **Import Mesh** dialog box appears.
2. Apply the following settings

Setting	Value
Files of type	PATRAN Neutral
Options > Mesh Units	m
File name	WingSPSMesh.out

3. Click **Open**.
4. To best orient the view, right-click a blank area in the viewer and select **Predefined Camera** > **Isometric View (Y up)** from the shortcut menu.

10.6.2. Creating the Domain

1. Edit `Case Options` > `General` in the **Outline** tree view and ensure that **Automatic Default Domain** is turned on. A domain named `Default Domain` should now appear under the `Simulation` branch.
2. Double-click `Default Domain` and apply the following settings

Tab	Setting	Value
Basic Settings	Location and Type > Location	WING ELEMENTS
	Fluid and Particle Definitions	Fluid 1
	Fluid and Particle Definitions > Fluid 1 > Material	Air Ideal Gas
	Domain Models > Pressure > Reference Pressure [*1* (p. 166)]	1 [atm]

Tab	Setting	Value
Fluid Models	Heat Transfer > Option	Total Energy [2 (p. 166)]
	Turbulence > Option	Shear Stress Transport

Footnotes

1. When using an ideal gas, it is important to set an appropriate reference pressure because some properties depend on the absolute pressure level.
2. The Total Energy model is appropriate for high-speed flows because it includes kinetic energy effects.

3. Click **OK**.

10.6.3. Creating the Boundaries

10.6.3.1. Creating an Inlet Boundary

1. Create a new boundary named `Inlet`.
2. Apply the following settings

Tab	Setting	Value
Basic Settings	Boundary Type	Inlet
	Location	INLET
Boundary Details	Flow Regime > Option	Supersonic
	Mass And Momentum > Option	Cart. Vel. & Pressure
	Mass And Momentum > U	600 [m s^-1]
	Mass And Momentum > V	0 [m s^-1]
	Mass And Momentum > W	0 [m s^-1]
	Mass And Momentum > Rel. Static Pres.	0 [Pa]
	Turbulence > Option	Intensity and Length Scale
	Turbulence > Fractional Intensity	0.01
	Turbulence > Eddy Length Scale	0.02 [m]
	Heat Transfer > Static Temperature	300 [K]

3. Click **OK**.

10.6.3.2. Creating an Outlet Boundary

1. Create a new boundary named `Outlet`.
2. Apply the following settings

Tab	Setting	Value
Basic Settings	Boundary Type	Outlet
	Location	OUTLET
Boundary Details	Flow Regime > Option	Supersonic

3. Click **OK**.

10.6.3.3. Creating the Symmetry Plane Boundaries

1. Create a new boundary named `SymP1`.
2. Apply the following settings

Tab	Setting	Value
Basic Settings	Boundary Type	Symmetry [1 (p. 167)]
	Location	SIDE1

Footnote

1. Symmetry, which can make a 3D problem into a 2D problem, can be used when geometry and mesh are invariant normal to the symmetry surface.

3. Click **OK**.
4. Create a new boundary named `SymP2`.
5. Apply the following settings

Tab	Setting	Value
Basic Settings	Boundary Type	Symmetry
	Location	SIDE2

6. Click **OK**.
7. Create a new boundary named `Bottom`.
8. Apply the following settings

Tab	Setting	Value
Basic Settings	Boundary Type	Symmetry
	Location	BOTTOM

9. Click **OK**.

10.6.3.4. Creating a Free Slip Boundary

1. Create a new boundary named `Top`.
2. Apply the following settings

Tab	Setting	Value
Basic Settings	Boundary Type	Wall
	Location	TOP
Boundary Details	Mass And Momentum > Option	Free Slip Wall

3. Click **OK**.

10.6.3.5. Creating a Wall Boundary

1. Create a new boundary named `WingSurface`.
2. Apply the following settings

Tab	Setting	Value
Basic Settings	Boundary Type	Wall
	Location	WING_Nodes [1 (p. 168)]

Footnote

1. If particular items do not appear in the drop-down list, click the *Ellipsis* ... icon to see all available items.

3. Click **OK**.

10.6.4. Creating Domain Interfaces

The imported mesh contains three regions that will be connected with domain interfaces.

1. Create a new domain interface named `Domain Interface 1`.
2. Apply the following settings

Tab	Setting	Value
Basic Settings	Interface Type	Fluid Fluid
	Interface Side 1 > Region List	Primitive 2D A [1 (p. 168)]
	Interface Side 2 > Region List	Primitive 2D, Primitive 2D B

Footnote

1. If particular items do not appear in the drop-down list, click the *Ellipsis* ... icon to see all available items.

3. Click **OK**.

10.6.5. Setting Initial Values

For high-speed compressible flow, the CFX-Solver usually requires sensible initial conditions to be set for the velocity field.

1. Click *Global Initialization*  .
2. Apply the following settings: [1]

Tab	Setting	Value
Global Settings	Initial Conditions > Cartesian Velocity Components > Option	Automatic with Value
	Initial Conditions > Cartesian Velocity Components > U	600 [m s^-1]
	Initial Conditions > Cartesian Velocity Components > V	0 [m s^-1]
	Initial Conditions > Cartesian Velocity Components > W	0 [m s^-1]
	Initial Conditions > Temperature > Option	Automatic with Value
	Initial Conditions > Temperature > Temperature	300 [K]

3. Click **OK**.

10.6.6. Setting the Solver Controls

The residence time for the fluid is the length of the domain divided by the speed of the liquid; using values from the problem specification, the result is approximately:

70 [m] / 600 [m s^-1] = 0.117 [s]

In the next step, you will set a maximum timescale, then the solver will start with a conservative time scale that gradually increases towards the fluid-residence time as the residuals decrease.

1. Click *Solver Control*  .
2. Apply the following settings

Tab	Setting	Value
Basic Settings	Convergence Control > Fluid Timescale Control > Maximum Timescale	(Selected)
	Convergence Control > Fluid Timescale Control > Maximum Timescale > Maximum Timescale	0.1 [s]
	Convergence Criteria > Residual Target	1.0e-05

[1] The values are from the problem specification.

3. Click **OK**.

10.6.7. Writing the CFX-Solver Input (.def) File

1. Click *Define Run*  .
2. Apply the following settings

Setting	Value
File name	WingSPS.def

3. Click **Save**.

   CFX-Solver Manager automatically starts and, on the **Define Run** dialog box, the **Solver Input File** is set.
4. If using Standalone Mode, quit CFX-Pre, saving the simulation (`.cfx`) file at your discretion.

10.7. Obtaining a Solution Using CFX-Solver Manager

At this point, CFX-Solver Manager is running, and the **Define Run** dialog box is displayed, with the CFX-Solver input file set.

1. Click **Start Run**.
2. Select the check box next to **Post-Process Results** when the completion message appears at the end of the run.
3. If using Standalone Mode, select the check box next to **Shut down CFX-Solver Manager**.
4. Click **OK**.

10.8. Viewing the Results in CFD-Post

The following topics will be discussed:

10.8.1. Displaying Mach Information

The first view configured shows that the bulk of the flow over the wing has a Mach Number of over 1.5.

1. To best orient the view, select **View From -Z** by typing **Shift** +**Z**.
2. Zoom in so the geometry fills the Viewer.
3. Create a new contour named `SymP2Mach`.
4. Apply the following settings

Tab	Setting	Value
Geometry	Locations	SymP2

Tab	Setting	Value
	Variable	Mach Number
	Range	User Specified
	Min	1
	Max	2
	# of Contours	21

5. Click **Apply**.
6. Clear the check box next to SymP2Mach.

10.8.2. Displaying Pressure Information

To display pressure information, create a contour plot that shows the pressure field:

1. Create a new contour named SymP2Pressure.
2. Apply the following settings

Tab	Setting	Value
Geometry	Locations	SymP2
	Variable	Pressure
	Range	Global

3. Click **Apply**.
4. Clear the check box next to SymP2Pressure.

10.8.3. Displaying Temperature Information

You can confirm that a significant energy loss occurs around the wing's leading edge by plotting temperature on SymP2.

1. Create a new contour named SymP2Temperature.
2. Apply the following settings

Tab	Setting	Value
Geometry	Locations	SymP2
	Variable	Temperature
	Range	Global

3. Click **Apply**.

   The contour shows that the temperature at the wing's leading edge is approximately 180 K higher than the inlet temperature.
4. Clear the check box next to SymP2Temperature.

10.8.4. Displaying Pressure With User Vectors

You can also create a user vector to show the pressure acting on the wing:

1. Create a new variable named `Variable 1`.
2. Apply the following settings

Name	Setting	Value
Variable 1	Vector	(Selected)
	X Expression	(Pressure+101325[Pa])*Normal X
	Y Expression	(Pressure+101325[Pa])*Normal Y
	Z Expression	(Pressure+101325[Pa])*Normal Z

3. Click **Apply**.
4. Create a new vector named `Vector 1`.
5. Apply the following settings

Tab	Setting	Value
Geometry	Locations	WingSurface
	Variable	Variable 1
Symbol	Symbol Size	0.04

6. Click **Apply**.
7. Zoom in on the wing in order to see the created vector plot.

Chapter 11: Flow Through a Butterfly Valve

This tutorial includes:

11.1. Tutorial Features

In this tutorial you will learn about:

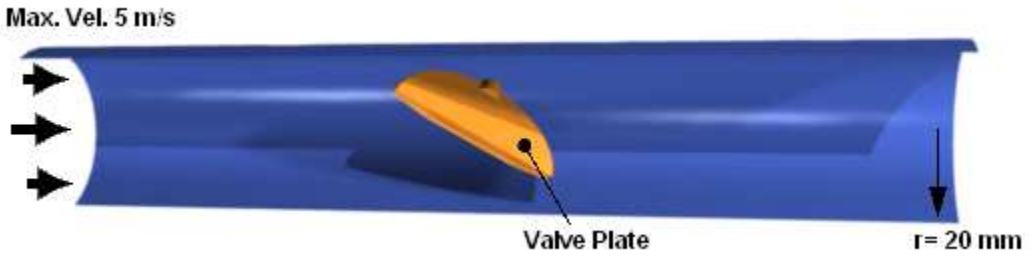
- Using a rough wall boundary in CFX-Pre to simulate the pipe wall
- Creating a fully developed inlet velocity profile using the CFX Expression Language
- Setting up a Particle Tracking simulation in CFX-Pre to trace sand particles
- Animating particle tracks in CFD-Post to trace sand particles through the domain
- Performing quantitative calculation of average static pressure in CFD-Post on the outlet boundary.

Component	Feature	Details
CFX-Pre	User Mode	General Mode
	Analysis Type	Steady State
	Fluid Type	General Fluid
	Domain Type	Single Domain
	Turbulence Model	k-Epsilon
	Heat Transfer	None
	Particle Tracking	
	Boundary Conditions	Inlet (Profile)
		Inlet (Subsonic)
		Outlet (Subsonic)
		Symmetry Plane
		Wall: No-Slip
		Wall: Rough
	CEL (CFX Expression Language)	
	Timestep	Auto Time Scale
CFD-Post	Plots	Animation

Component	Feature	Details
		Default Locators
		Particle Track
		Point
		Slice Plane
	Other	Changing the Color Range
		Movie Generation
		Particle Track Animation
		Quantitative Calculation
		Symmetry, Reflection Plane

11.2. Overview of the Problem to Solve

Pumps and compressors are commonplace. An estimate of the pumping requirement can be calculated based on the height difference between source and destination and head loss estimates for the pipe and any obstructions/joints along the way. Investigating the detailed flow pattern around a valve or joint however, can lead to a better understanding of why these losses occur. Improvements in valve/joint design can be simulated using CFD, and implemented to reduce pumping requirements and cost.



Flows can contain particulates that affect the flow and cause erosion to pipe and valve components. You can use the particle-tracking capability of CFX to simulate these effects.

In this example, water flows at 5 m/s through a 20 mm radius pipe that has a rough internal surface. The velocity profile is assumed to be fully developed at the pipe inlet. The flow, which is controlled by a butterfly valve set at an angle of 55° to the vertical axis, contains sand particles ranging in size from 50 to 500 microns. The equivalent sand grain roughness is 0.2 mm.

The reference temperature is 300 K; the reference pressure is 1 atm.

11.3. Approach to the Problem

A mesh is provided. You will create sand particles and a domain that contains water; for one part of the simulation the water and sand will be fully coupled, and for the other part of the simulation they will be one-way coupled. To increase the accuracy of the simulation, the inlet will be given a velocity profile that simulates a fully-developed boundary layer.

To solve the simulation, you will create two sets of identical particles. The first set will be fully coupled to predict the effect of the particles on the continuous phase flow field and allow the particles to influence the flow field. The second set will be one-way coupled but will contain a much higher number of particles

to provide a more accurate calculation of the particle volume fraction and local forces on walls, but without affecting the flow field.

11.4. Before You Begin

If this is the first tutorial you are working with, it is important to review the following topics before beginning:

- *Preparing a Working Directory and Starting ANSYS CFX in Standalone Mode* (p. 1)
- *Running ANSYS CFX Tutorials in ANSYS Workbench* (p. 2)
- *Changing the Display Colors* (p. 5)
- *Playing a Tutorial Session File* (p. 4)

11.5. Starting CFX-Pre

1. Prepare the working directory using the following files in the `examples` directory:
   - `PipeValveMesh.gtm`
   - `PipeValve.pre`
2. Set the working directory and start CFX-Pre.

   For details, see *Preparing a Working Directory and Starting ANSYS CFX in Standalone Mode* (p. 1)

11.6. Defining a Simulation in CFX-Pre

If you want to set up the simulation automatically using a tutorial session file, run `PipeValve.pre`. For details, see *Playing a Tutorial Session File* (p. 4). Then proceed to *Obtaining a Solution Using CFX-Solver Manager* (p. 186).

If you want to set up the simulation manually, proceed to *Creating a New Case* (p. 175)

11.6.1. Creating a New Case

1. In CFX-Pre, select **File** > **New Case**.
2. Select **General** and click **OK**.
3. Select **File** > **Save Case As**.
4. Under **File name**, type `PipeValve`.
5. Click **Save**.

11.6.2. Importing the Mesh

1. Right-click `Mesh` and select **Import Mesh** > **CFX Mesh**. The **Import Mesh** dialog box appears.
2. Apply the following settings:

Setting	Value
File name	PipeValveMesh.gtm

3. Click **Open**.

11.6.3. Defining the Properties of the Sand

The material properties of the sand particles used in the simulation need to be defined. Heat transfer and radiation modeling are not used in this simulation, so the only properties that need to be defined are the density of the sand and the diameter range.

To calculate the effect of the particles on the continuous fluid, between 100 and 1000 particles are usually required. However, if accurate information about the particle volume fraction or local forces on wall boundaries is required, then a much larger number of particles needs to be modeled.

When you create the domain, choose either full coupling or one-way coupling between the particle and continuous phase. *Full coupling* is needed to predict the effect of the particles on the continuous phase flow field but has a higher CPU cost than one-way coupling; *one-way coupling* simply predicts the particle paths during post-processing based on the flow field, but without affecting the flow field.

To optimize CPU usage, you can create two sets of identical particles. The first set should be fully coupled and around 200 particles will be used. This allows the particles to influence the flow field. The second set uses one-way coupling but contains 5000 particles. This provides a more accurate calculation of the particle volume fraction and local forces on walls. (These values are defined in the inlet boundary definition.)

For this tutorial you will create a "Sand Fully Coupled" boundary condition that has 200 particles moving with a mass flow rate of 0.01 kg/s and a "Sand One Way Coupled" boundary condition that has 5000 particles moving with a mass flow rate of 0.01 kg/s. In both cases the sand density is 2300 [kg m^-3]; particle diameters range from 50 e-6 m. to 500 e-6 m. with an average diameter of 250 e-6 m. and a standard deviation of 70 e-6 m. You will set a Finnie erosion model with a velocity power factor of 2 and a reference velocity of 1 m/s.

1. Click *Insert Material*  then create a new material named `Sand Fully Coupled`.
2. Apply the following settings:

Tab	Setting	Value
Basic Settings	Material Group	Particle Solids
	Thermodynamic State	(Selected)
Material Properties	Thermodynamic Properties > Equation of State > Density	2300 [kg m^-3] [1 (p. 177)]
	Thermodynamic Properties > Specific Heat Capacity	(Selected)
	Thermodynamic Properties > Specific Heat Capacity > Specific Heat Capacity	0 [J kg^-1 K^-1] [2 (p. 177)]
	Thermodynamic Properties > Reference State	(Selected)
	Thermodynamic Properties > Reference State > Option	Specified Point
	Thermodynamic Properties > Reference State > Ref. Temperature	300 [K] [1 (p. 177)]

Footnotes

1. From the problem description.
2. This value is not used because heat transfer is not modeled in this tutorial.

3. Click **OK**.
4. Under `Materials`, right-click `Sand Fully Coupled` and select **Duplicate** from the shortcut menu.
5. Rename the duplicate as `Sand One Way Coupled`.
6. `Sand One Way Coupled` is created with properties identical to `Sand Fully Coupled`.

11.6.4. Creating the Domain

Set up an environment that has water and sand defined in two ways; one in which the sand is fully coupled, and one in which the sand is one-way coupled:

1. Edit `Case Options` > `General` in the **Outline** tree view and ensure that **Automatic Default Domain** is turned on. A domain named `Default Domain` should appear under the `Simulation` > `Flow Analysis 1` branch.
2. Double-click `Default Domain`.
3. Under **Fluid and Particle Definitions**, delete `Fluid 1` and click *Add new item*  to create three new materials named Water, Sand Fully Coupled, and Sand One Way Coupled.
4. Apply the following settings, which are taken from the problem description:

Tab	Setting	Value
Basic Settings	Fluid and Particle Definitions	Water
	Fluid and Particle Definitions > Water > Material	Water
	Fluid and Particle Definitions	Sand Fully Coupled
	Fluid and Particle Definitions > Sand Fully Coupled > Material	Sand Fully Coupled [*1* (p. 179)]
	Fluid and Particle Definitions > Sand Fully Coupled > Morphology > Option	Particle Transport Solid
	Fluid and Particle Definitions > Sand Fully Coupled > Morphology > Particle Diameter Distribution	(Selected)
	Fluid and Particle Definitions > Sand Fully Coupled > Morphology > Particle Diameter Distribution > Option	Normal in Diameter by Mass
	Fluid and Particle Definitions > Sand Fully Coupled > Morphology > Particle Diameter Distribution > Minimum Diameter	50e-6 [m]
	Fluid and Particle Definitions > Sand Fully Coupled > Morphology > Particle Diameter Distribution > Maximum Diameter	500e-6 [m]

Tab	Setting	Value
	Fluid and Particle Definitions > Sand Fully Coupled > Morphology > Particle Diameter Distribution > Mean Diameter	250e-6 [m]
	Fluid and Particle Definitions > Sand Fully Coupled > Morphology > Particle Diameter Distribution > Std. Deviation	70e-6 [m]
	Fluid and Particle Definitions	Sand One Way Coupled
	Fluid and Particle Definitions > Sand One Way Coupled > Material	Sand One Way Coupled [2 (p. 179)]
	Fluid and Particle Definitions > Sand One Way Coupled > Morphology > Option	Particle Transport Solid
	Fluid and Particle Definitions > Sand One Way Coupled > Morphology > Particle Diameter Distribution	(Selected)
	Fluid and Particle Definitions > Sand One Way Coupled > Morphology > Particle Diameter Distribution > Option	Normal in Diameter by Mass
	Fluid and Particle Definitions > Sand One Way Coupled > Morphology > Particle Diameter Distribution > Minimum Diameter	50e-6 [m]
	Fluid and Particle Definitions > Sand One Way Coupled > Morphology > Particle Diameter Distribution > Maximum Diameter	500e-6 [m]
	Fluid and Particle Definitions > Sand One Way Coupled > Morphology > Particle Diameter Distribution > Mean Diameter	250e-6 [m]
	Fluid and Particle Definitions > Sand One Way Coupled > Morphology > Particle Diameter Distribution > Std. Deviation	70e-6 [m]
	Domain Models > Pressure > Reference Pressure	1 [atm]
Fluid Models	Heat Transfer > Option	None
	Turbulence > Option	k-Epsilon [3 (p. 179)]
Fluid Specific Models	Fluid	Sand Fully Coupled
	Fluid > Sand Fully Coupled > Erosion Model > Option	Finnie
	Fluid > Sand Fully Coupled > Erosion Model > Vel. Power Factor	2.0
	Fluid > Sand Fully Coupled > Erosion Model > Reference Velocity	1 [m s^-1]
	Fluid > Sand One Way Coupled	(Selected)

Tab	Setting	Value
	Fluid > Sand One Way Coupled > Erosion Model > Option	Finnie
	Fluid > Sand One Way Coupled > Erosion Model > Vel. Power Factor	2.0
	Fluid > Sand One Way Coupled > Erosion Model > Reference Velocity	1 [m s^-1]

Footnotes

1. Click the *Ellipsis*  icon to open the **Materials** dialog box, then select `Particle Solids` > `Sand Fully Coupled`.
2. Click the *Ellipsis*  icon to open the **Materials** dialog box, then select `Particle Solids` > `Sand One Way Coupled`.
3. The turbulence model applies only to the continuous phase and not the particle phases.

5. Apply the following settings:

Tab	Setting	Value
Fluid Pair Models	Fluid Pair	Water | Sand Fully Coupled
	Fluid Pairs > Water | Sand Fully Coupled > Particle Coupling	Fully Coupled
	Fluid Pairs >Water | Sand Fully Coupled > Momentum Transfer > Drag Force > Option	Schiller Naumann [1 (p. 179)]
	Fluid Pair	Water | Sand One Way Coupled
	Fluid Pairs > Water | Sand One Way Coupled > Particle Coupling	One-way Coupling
	Fluid Pairs > Water | Sand One Way Coupled > Momentum Transfer > Drag Force > Option	Schiller Naumann

Footnote

1. The Schiller Naumann drag model is appropriate for sparsely-distributed, solid spherical particles.

6. Click **OK**.

11.6.5. Creating the Inlet Velocity Profile

In previous tutorials you have often defined a uniform velocity profile at an inlet boundary. This means that the inlet velocity near to the walls is the same as that at the center of the inlet. If you look at the results from these simulations, you will see that downstream of the inlet a boundary layer will develop, so that the downstream near wall velocity is much lower than the inlet near wall velocity.

You can simulate an inlet more accurately by defining an inlet velocity profile, so that the boundary layer is already fully developed at the inlet. The one seventh power law will be used in this tutorial to describe the profile at the pipe inlet. The equation for this is:

$$U = W_{\max}\left(1 - \frac{r}{R_{\max}}\right)^{\frac{1}{7}} \qquad \textbf{(11–1)}$$

where $W_{\max}$ is the pipe centerline velocity, $R_{\max}$ is the pipe radius, and r is the distance from the pipe centerline.

You can create a non-uniform (profile) boundary condition by doing one of the following:

- Creating an expression using CEL that describes the inlet profile. Using a CEL expression is the easiest way to create the profile.
- Creating a User CEL Function that uses a user subroutine (linked to the CFX-Solver during execution) to describe the inlet profile. The User CEL Function method is more complex, but is provided here as an example of how to use this feature.
- Loading a BC profile file (a file that contains boundary condition profile data).

  Profiles created from data files are not used in this tutorial, but are used in the tutorial *Flow in a Process Injection Mixing Pipe* (p. 79).

Note

For complex profiles, it may be necessary to use a User CEL Function or a BC profile file.

Use a CEL expression to define the velocity profile for the inlet boundary:

1. Click *Insert Expression* and create the following expressions using *Equation 11–1* (p. 180) and values from the problem description:

Name	Definition
Rmax	20 [mm]
Wmax	5 [m s^-1]
Wprof	Wmax*(abs(1-r/Rmax)^0.143)

   In the definition of `Wprof`, the variable r (radius) is a CFX System Variable defined as:

$$r = \sqrt{x^2 + y^2} \quad \textbf{(11–2)}$$

In this equation, x and y are defined as directions 1 and 2 (X and Y for Cartesian coordinate frames) respectively, in the selected reference coordinate frame.

2. Continue with the tutorial at *Creating the Boundary Conditions* (p. 181).

11.6.6. Creating the Boundary Conditions

11.6.6.1. Inlet Boundary

1. Create a new boundary named `inlet`.
2. Apply the following settings, using values from the problem description:

Tab	Setting	Value
Basic Settings	Boundary Type	Inlet
	Location	inlet
Boundary Details	Mass And Momentum > Option	Cart. Vel. Components
	Mass And Momentum > U	0 [m s^-1]
	Mass And Momentum > V	0 [m s^-1]
	Mass And Momentum > W	Wprof [1 (p. 182)]
Fluid Values [2 (p. 182)]	Boundary Conditions	Sand Fully Coupled
	Boundary Conditions > Sand Fully Coupled > Particle Behavior > Define Particle Behavior	(Selected)
	Boundary Conditions > Sand Fully Coupled > Mass and Momentum > Option	Cart. Vel. Components [3 (p. 182)]
	Boundary Conditions > Sand Fully Coupled > Mass And Momentum > U	0 [m s^-1]
	Boundary Conditions > Sand Fully Coupled > Mass And Momentum > V	0 [m s^-1]
	Boundary Conditions > Sand Fully Coupled > Mass And Momentum > W	Wprof [4 (p. 182)]
	Boundary Conditions > Sand Fully Coupled > Particle Position > Option	Uniform Injection
	Boundary Conditions > Sand Fully Coupled > Particle Position > Number of Positions > Option	Direct Specification
	Boundary Conditions > Sand Fully Coupled > Particle Position > Number of Positions > Number	200
	Boundary Conditions > Sand Fully Coupled > Particle Mass Flow > Mass Flow Rate	0.01 [kg s^-1]

Tab	Setting	Value
	Boundary Conditions	Sand One Way Coupled
	Boundary Conditions > Sand One Way Coupled > Particle Behavior > Define Particle Behavior	(Selected)
	Boundary Conditions > Sand One Way Coupled > Mass and Momentum > Option	Cart. Vel. Components [3 (p. 182)]
	Boundary Conditions > Sand One Way Coupled > Mass And Momentum > U	0 [m s^-1]
	Boundary Conditions > Sand One Way Coupled > Mass And Momentum > V	0 [m s^-1]
	Boundary Conditions > Sand One Way Coupled > Mass And Momentum > W	Wprof [4 (p. 182)]
	Boundary Conditions > Sand One Way Coupled > Particle Position > Option	Uniform Injection
	Boundary Conditions > Sand One Way Coupled > Particle Position > Number of Positions > Option	Direct Specification
	Boundary Conditions > Sand One Way Coupled > Particle Position > Number of Positions > Number	5000
	Boundary Conditions > Sand One Way Coupled > Particle Position > Particle Mass Flow Rate > Mass Flow Rate	0.01 [kg s^-1]

Footnotes

1. Use the **Expressions** details view  to enter `Wprof`.
2. Do *not* select **Particle Diameter Distribution**. The diameter distribution was defined when creating the domain; this option would override those settings for this boundary only.
3. Instead of manually specifying the same velocity profile as the fluid, you can also select the Zero Slip Velocity option.
4. As you did on the Boundary Details tab.

3. Click **OK**.

One-way coupled particles are tracked as a function of the fluid flow field. The latter is not influenced by the one-way coupled particles. The fluid flow will therefore be influenced by the 0.01 [kg s^-1] flow of two-way coupled particles, but not by the 0.01 [kg s^-1] flow of one-way coupled particles.

11.6.6.2. Outlet Boundary

1. Create a new boundary named `outlet`.

2. Apply the following settings:

Tab	Setting	Value
Basic Settings	Boundary Type	Outlet
	Location	outlet
Boundary Details	Flow Regime > Option	Subsonic
	Mass and Momentum > Option	Average Static Pressure
	Mass and Momentum > Relative Pressure	0 [Pa]

3. Click **OK**.

11.6.6.3. Symmetry Plane Boundary

1. Create a new boundary named `symP`.
2. Apply the following settings:

Tab	Setting	Value
Basic Settings	Boundary Type	Symmetry [1 (p. 183)]
	Location	symP

Footnote

1. Symmetry can be used when geometry and mesh are invariant normal to the symmetry surface.

3. Click **OK**.

11.6.6.4. Pipe Wall Boundary

1. Create a new boundary named `pipe wall`.
2. Apply the following settings:

Tab	Setting	Value
Basic Settings	Boundary Type	Wall
	Location	pipe wall
Boundary Details	Wall Roughness > Option	Rough Wall
	Wall Roughness > Sand Grain Roughness	0.2 [mm] [1 (p. 184)]
Fluid Values	Boundary Conditions	Sand Fully Coupled
	Boundary Conditions > Sand Fully Coupled > Velocity > Option	Restitution Coefficient
	Boundary Conditions > Sand Fully Coupled > Velocity > Perpendicular Coeff.	0.8 [2 (p. 184)]

Tab	Setting	Value
	Boundary Conditions > Sand Fully Coupled > Velocity > Parallel Coeff.	1

Footnotes

1. From the problem description. Make sure that you change the units to millimeters. The thickness of the first element should be of the same order as the roughness height.
2. This value would typically come from experimental or reference data.

3. Apply the same setting values for `Sand One Way Coupled` as for `Sand Fully Coupled`.
4. Click **OK**.

11.6.6.5. Editing the Default Boundary

1. In the **Outline** tree view, edit the boundary named `Default Domain Default`.
2. Apply the following settings:

Tab	Setting	Value
Fluid Values	Boundary Conditions	Sand Fully Coupled
	Boundary Conditions > Sand Fully Coupled > Velocity > Perpendicular Coeff.	0.9 [1 (p. 184)]
	Boundary Conditions	Sand One Way Coupled
	Boundary Conditions > Sand One Way Coupled > Velocity > Perpendicular Coeff.	0.9

Footnote

1. This value would typically come from experimental or reference data. For this tutorial, the pipe wall and butterfly valve are considered to be made of different materials, so their perpendicular coefficients are different.

3. Click **OK**.

11.6.7. Setting Initial Values

Set up the initial values to be consistent with the inlet boundary conditions:

1. Click *Global Initialization*  .
2. Apply the following settings:

Tab	Setting	Value
Global Settings	Initial Conditions > Cartesian Velocity Components > Option	Automatic with Value
	Initial Conditions > Cartesian Velocity Components > Option > U	0 [m s^-1]
	Initial Conditions > Cartesian Velocity Components > Option > V	0 [m s^-1]
	Initial Conditions > Cartesian Velocity Components > Option > W	Wprof

3. Click **OK**.

11.6.8. Setting the Solver Controls

1. Click *Solver Control*  .
2. Apply the following settings:

Tab	Setting	Value
Basic Settings	Advection Scheme > Option	High Resolution
Particle Control	Particle Integration > Max. Particle Intg. Time Step	(Selected)
	Particle Integration > Max. Particle Intg. Time Step > Value	1e+10 [s]
	Particle Termination Control	(Selected)
	Particle Termination Control > Maximum Tracking Time	(Selected)
	Particle Termination Control > Maximum Tracking Time > Value	10 [s]
	Particle Termination Control > Maximum Tracking Distance	(Selected)
	Particle Termination Control > Maximum Tracking Distance > Value	10 [m]
	Particle Termination Control > Max. Num. Integration Steps	(Selected)
	Particle Termination Control > Max. Num. Integration Steps > Value	10000 [1 (p. 185)]

Footnote

1. This value controls the number of mesh elements a particle is allowed to cross and therefore must take into account the size and density of the mesh.

Note

The numeric values in the preceding table are all designed to put a high upper limit on the amount of processing that will be done. For example, the tracking time of 10 seconds would allow a particle to get caught in an eddy for a reasonable amount of time.

3. Click **OK**.

11.6.9. Writing the CFX-Solver Input (.def) File

1. Click *Define Run*  .
2. Apply the following settings:

Setting	Value
File name	`PipeValve.def`

3. Click **Save**.

   CFX-Solver Manager automatically starts and, on the **Define Run** dialog box, the **Solver Input File** is set.
4. If using Standalone Mode, quit CFX-Pre, saving the simulation (`.cfx`) file at your discretion.

11.7. Obtaining a Solution Using CFX-Solver Manager

When CFX-Pre has shut down and CFX-Solver Manager has started, you can obtain a solution to the CFD problem by using the procedure that follows.

1. Ensure the **Define Run** dialog box is displayed and click **Start Run**.
2. Select the check box next to **Post-Process Results** when the completion message appears at the end of the run.
3. If using Standalone Mode, select the check box next to **Shut down CFX-Solver Manager**.
4. Click **OK**.

11.8. Viewing the Results in CFD-Post

In this section, you will first plot erosion on the valve surface and side walls due to the sand particles. You will then create an animation of particle tracks through the domain.

11.8.1. Erosion Due to Sand Particles

An important consideration in this simulation is erosion to the pipe wall and valve due to the sand particles. A good indication of erosion is given by the `Erosion Rate Density` parameter, which corresponds to pressure and shear stress due to the flow.

1. Edit the object named `Default Domain Default`.
2. Apply the following settings using the *Ellipsis*  as required for variable selection

Tab	Setting	Value
Color	Mode	Variable
	Variable	Sand One Way Coupled.Erosion Rate Density [1 (p. 187)]
	Range	User Specified
	Min	0 [kg m^-2 s^-1]
	Max	25 [kg m^-2 s^-1] [2 (p. 187)]

Footnotes

1. This is statistically better than `Sand Fully Coupled.Erosion Rate Density` because many more particles were calculated for `Sand One Way Coupled`.
2. This range is used to gain a better resolution of the wall shear stress values around the edge of the valve surfaces.

3. Click **Apply**.

   As can be seen, the highest values occur on the edges of the valve where most particles strike. Erosion of the low Z side of the valve would occur more quickly than for the high Z side.

11.8.2. Displaying Erosion on the Pipe Wall

Set the user specified range for coloring to resolve areas of stress on the pipe wall near of the valve:

1. Ensure that the check box next to `Res PT for Sand Fully Coupled` is cleared.
2. Ensure that the check box next to `Default Domain Default` is cleared.
3. Edit the object named `pipe wall`.
4. Apply the following settings:

Tab	Setting	Value
Color	Mode	Variable
	Variable	Sand One Way Coupled.Erosion Rate Density
	Range	User Specified
	Min	0 [kg m^-2 s^-1]
	Max	25 [kg m^-2 s^-1]

5. Click **Apply**.
6. Optionally, fill the check box next to `Default Domain Default` to see how sand particles have deflected off the butterfly valve then to the pipe wall.

11.8.3. Setting the Particle Tracks

Default particle track objects are created at the start of the session. One particle track is created for each set of particles in the simulation. You are going to make use of the default object for `Sand Fully Coupled`.

The default object draws 25 tracks as lines from the inlet to outlet. The **Info** tab shows information about the total number of tracks, the index range, and the track numbers that are drawn.

1. Edit the object named `Res PT for Sand Fully Coupled`.
2. Apply the following settings:

Tab	Setting	Value
Geometry	Max Tracks	20 [1 (p. 188)]

Footnote

1. This value improves the resolution of the tracks generated.

3. Click **Apply**.
4. Right-click on a blank area anywhere in the viewer, select **Predefined Camera** from the shortcut menu and select **View From +X** to view the particle track.

11.8.4. Creating the Particle Track Symbols

1. Turn off the visibility for all objects except `Wireframe`.
2. Edit the object named `Res PT for Sand Fully Coupled`.
3. Apply the following settings:

Tab	Setting	Value
Color	Mode	Variable
	Variable	Sand Fully Coupled.Velocity w
Symbol	Show Symbols	(Selected)
	Show Symbols > Max Time	0 [s]
	Show Symbols > Min Time	0 [s]
	Show Symbols > Interval	0.07 [s]
	Show Symbols > Symbol	Ball
	Show Symbols > Scale	1.2

4. Clear **Show Tracks**.
5. Click **Apply**.

Symbols are placed at the start of each track.

11.8.5. Creating a Particle Track Animation

The following steps describe how to create a particle tracking animation using **Quick Animation**. Similar effects can be achieved in more detail using the **Keyframe Animation** option, which allows full control over all aspects on an animation.

1. Right-click a blank area in the viewer and select **Predefined Camera** > **Isometric View (Y up)** from the shortcut menu.
2. Right-click an edge of the flat side on the half cylinder and select **Reflect/Mirror** from the shortcut menu. Click **X Axis** to choose it as the normal direction.

   Note

   Alternatively, you can apply **Reflect/Mirror**, by double-clicking `Default Domain` to open up the **Details** panel. In the **Instancing** tab enable **Apply Reflection** and select **Method** to `YZ Plane`. Click **Apply**.

3. Select **Tools** > **Animation** or click *Animation*  .
4. Select **Quick Animation**.
5. Select `Res PT for Sand Fully Coupled`:
6. Click **Options** to display the **Animation Options** dialog box, then clear **Override Symbol Settings** to ensure the symbol type and size are kept at their specified settings for the animation playback. Click **OK**.

   Note

   The arrow pointing downward in the bottom right corner of the Animation Window will reveal the **Options** button if it is not immediately visible.

7. Select **Loop**.
8. Clear *Repeat forever*  and ensure **Repeat** is set to `1`.
9. Select **Save Movie**.
10. Set **Format** to `MPEG1`.
11. Click *Browse*  and enter `tracks.mpg` as the file name.
12. Click *Play the animation*  .
13. If prompted to overwrite an existing movie, click **Overwrite**.

    The animation plays and builds an `.mpg` file.
14. Close the **Animation** dialog box.

11.8.6. Determining Minimum, Maximum, and Average Pressure Values

On the outlet boundary you created in CFX-Pre, you set the **Average Static Pressure** to `0.0 [Pa]`. To see the effect of this:

1. From the main menu select **Tools** > **Function Calculator**.

   The **Function Calculator** is displayed. It allows you to perform a wide range of quantitative calculations on your results.

   Note

   You should use Conservative variable values when performing calculations and Hybrid values for visualization purposes. Conservative values are set by default in CFD-Post but you can manually change the setting for each variable in the Variables Workspace, or the settings for all variables by using the Function Calculator. For details, see Hybrid and Conservative Variable Values in the *CFX Reference Guide*.

2. Set **Function** to `maxVal`.
3. Set **Location** to `outlet`.
4. Set **Variable** to `Pressure`.
5. Click **Calculate**.

   The result is the maximum value of pressure at the outlet.
6. Perform the calculation again using `minVal` to obtain the minimum pressure at the outlet.
7. Select `areaAve`, and then click **Calculate**.
   - This calculates the area weighted average of pressure.
   - The average pressure is approximately zero, as specified by the boundary.

11.8.6.1. Other Features

The geometry was created using a symmetry plane. In addition to the **Reflect/Mirror** option from the shortcut menu, you also can display the other half of the geometry by creating a YZ Plane at X = 0 and then editing the `Default Transform` object to use this plane as a reflection plane.

When you have finished viewing the results, quit CFD-Post.

Chapter 12: Flow in a Catalytic Converter

This tutorial includes:

12.1. Tutorial Features

In this tutorial you will learn about:

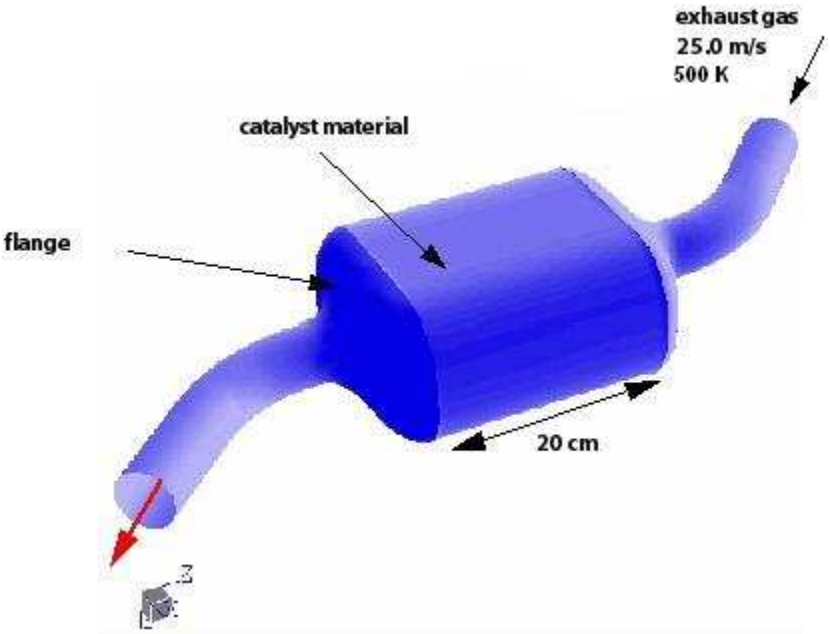
- Using multiple meshes in CFX-Pre.
- Joining meshes together using static fluid-fluid domain interfaces between the inlet/outlet flanges and the central catalyst body.
- Applying a source of resistance using a directional loss model.
- Creating a chart to show pressure drop versus Z coordinate in CFD-Post.
- Exporting data from a line locator to a file.

Component	Feature	Details
CFX-Pre	User Mode	General Mode
	Analysis Type	Steady State
	Domain Type	Multiple Domain (Fluid, Porous)
	Fluid Type	Ideal Gas
	Turbulence Model	k-Epsilon
	Heat Transfer	Thermal Energy
	Boundary Conditions	Inlet (Subsonic)
		Outlet (Subsonic)
		Wall: No-Slip
	Domain Interfaces	Fluid-Porous
	Timestep	Physical Time Scale
CFD-Post	Plots	Contour
		Default Locators
		Outline Plot (Wireframe)
		Polyline

Component	Feature	Details
		Slice Plane
		Vector
	Other	Chart Creation
		Data Export
		Title/Text
		Viewing the Mesh

12.2. Overview of the Problem to Solve

Catalytic converters are used on most vehicles on the road today. They reduce harmful emissions from internal combustion engines (such as oxides of nitrogen, hydrocarbons, and carbon monoxide) that are the result of incomplete combustion. Most new catalytic converters are the honeycomb ceramic type and are usually coated with platinum, rhodium, or palladium.



In this tutorial, you will model a catalytic converter in order to determine the pressure drop and heat transfer through it when air enters the inlet at 25 m/s and 500 K, and exits the outlet at a static pressure of 1 atm. For simplicity, you will not model chemical reactions.

You are provided with a mesh for the passageways inside a pipe-and-flange structure. You will use this mesh, and a copy of it, to model the pipe and flange portions of the flow field, at both ends of the catalytic converter.

For the housing, you are provided with a hexahedral mesh that was created in ICEM-Hexa. This mesh fills the entire 3D volume of the housing.

To model the presence of the honeycomb structure that exists in the housing, you will model porosity and apply resistance to the flow. The honeycomb structure has a porosity of 70%, which means that 70% of the total volume is available for fluid flow, while the other 30% is occupied by the solid material that comprises the honeycomb structure. The solid component of the structure will be steel. The honeycomb structure is lined up with the flow to prevent flow travel in the transverse direction. To model resistance to the flow,

you will apply a streamwise quadratic resistance coefficient of 650 kg m^-4. To reduce the amount of transverse flow, apply a quadratic resistance coefficient of 6500 kg m^-4 in the transverse direction. These given resistance coefficients are based on the superficial flow velocity, rather than the true flow velocity.

The Inlet boundary has a static temperature of 500 K. You will model heat transfer through the solid material in the porous domain. The heat transfer between the air and steel within the porous domain is modelled using an interfacial area density of 360 m^-1 and a heat transfer coefficient of 50 W m^-2 K^-1. Thermal energy is lost to the environment through the midsection walls of the catalytic converter; the rate of heat loss is defined by the heat transfer coefficient (20 W m^-2 K^-1) and the outside temperature (40 °C).

12.3. Approach to the Problem

You will first import the mesh for the housing. You will then import a mesh for one of the two flanges. You will then produce another flange mesh by transforming the first.

You will create one porous domain for the housing, and one fluid domain for both flanges.

You will model a honeycomb structure inside the housing by specifying a porosity and applying a directional momentum loss model.

12.4. Before You Begin

If this is the first tutorial you are working with, it is important to review the following topics before beginning:

- *Preparing a Working Directory and Starting ANSYS CFX in Standalone Mode* (p. 1)
- *Running ANSYS CFX Tutorials in ANSYS Workbench* (p. 2)
- *Changing the Display Colors* (p. 5)
- *Playing a Tutorial Session File* (p. 4)

12.5. Starting CFX-Pre

1. Prepare the working directory using the following files in the `examples` directory:
   - `CatConvHousing.hex`
   - `CatConvMesh.gtm`
   - `CatConv.pre`
   - `CatConv.ccl`
2. Set the working directory and start CFX-Pre.

   For details, see *Preparing a Working Directory and Starting ANSYS CFX in Standalone Mode* (p. 1)

12.6. Defining a Case in CFX-Pre

If you want to set up the simulation automatically using a tutorial session file, run `CatConv.pre`. For details, see *Playing a Tutorial Session File* (p. 4). Then proceed to *Obtaining a Solution Using CFX-Solver Manager* (p. 203).

If you want to set up the simulation manually, proceed to the following steps:

1. In CFX-Pre, select **File** > **New Case**.
2. Select **General** and click **OK**.

3. Select **File** > **Save Case As**.
4. Under **File name**, type `CatConv`.
5. Click **Save**.

12.6.1. Importing the Meshes and CCL File

The mesh for this catalytic converter comprises three distinct parts:

- The outlet section (pipe and flange).
- The inlet section (pipe and flange).

  You will copy and rotate the outlet section through 180 degrees to create this section.
- The catalyst (or monolith).

You will import a CFX Command Language (CCL) file defining mathematical expressions for this case. Next you will import the catalyst housing and a generic inlet/outlet section from provided files.

12.6.1.1. Importing the Required Expressions From a CCL File

The CCL file you are going to import contains expressions which will be used to define parameters in the simulation. These can be seen in the table below:

Expression	Value
AreaDen	360 [m^-1]
HTC	50 [W m^-2 K^-1]
HTCoutside	20 [W m^-2 K^-1]
L	0.4 [m]
Porosity	0.7
Tinlet	500[K]
Toutside	40 [C]

Import the CCL File to define relevant expressions:

1. Select **File** > **Import** > **CCL**.

   The **Import CCL** dialog box appears.
2. Select `CatConv.ccl`
3. Click **Open**.
4. Expand the `Expressions` section in the **Outline** tree to see a list of the expressions that have been imported.

12.6.1.2. Importing the Housing Mesh

The first mesh that you will import, `CatConvHousing.hex`, is a hexahedral mesh for the catalyst housing. The mesh was originally created in ICEM-Hexa using mesh units of centimeters. Because this type of mesh file does not specify the mesh units, you must specify them manually. The imported mesh has a width in the x-direction of 21 cm and a length in the z-direction of 20 cm.

1. Edit `Case Options` > `General` in the **Outline** tree view and ensure that **Automatic Default Domain** is turned off.

   Default Domain generation should be turned off because you will create a new domain manually, later in this tutorial.

2. Right-click `Mesh` and select **Import Mesh** > **Other**.

   The **Import Mesh** dialog box appears.

3. Apply the following settings:

Setting	Value
Files of type	All Types(*)
Mesh Format	ICEM CFD
Options > Mesh Units	cm
File name	CatConvHousing.hex

4. Click **Open**.

Later in this tutorial, you will create a porous domain for the housing in order to simulate flow through a honeycomb structure.

12.6.1.3. Importing the Pipe and Flange Mesh

The second mesh that you will import, `CatConvMesh.gtm`, is a mesh for a pipe and flange. The mesh has units of centimeters. Because this type of mesh file does specify the mesh units, there is no need to specify them manually.

1. Right-click `Mesh` and select **Import Mesh** > **CFX Mesh**.
2. Apply the following settings:

Setting	Value
File name	CatConvMesh.gtm

3. Click **Open**.

You now have a pipe and flange on the outlet end of the housing. In the next step, you will create a transformed copy of the pipe and flange for the inlet end.

12.6.1.4. Creating a Second Pipe and Flange Mesh

Create a transformed copy of the pipe and flange mesh:

1. Right-click `CatConvMesh.gtm` and select **Transform Mesh**.

   The **Mesh Transformation Editor** dialog box appears.

2. Apply the following settings:

Setting	Value
Transformation	Rotation

Setting	Value
Rotation Option	Rotation Axis
From	0, 0, 0.16
To	0, 1, 0.16 [1 (p. 196)]
Rotation Angle Option	Specified
Rotation Angle	180 [degree]
Multiple Copies	(Selected)
Multiple Copies > # of Copies	1

Footnote

1. This specifies an axis located at the center of the housing parallel to the y-axis.

3. Click **Apply**.

Later in this tutorial, you will create a fluid domain for both pipe and flange sections.

12.6.1.5. Creating a Single Region for Both Pipe and Flange Meshes

The outlet pipe and flange region is `B1.P3`. The inlet pipe and flange region is `B1.P3 2`.

There are three basic options for creating fluid domains on these regions:

- Create two similar domains: one that applies to `B1.P3`, and one that applies to `B1.P3 2`.
- Create one domain that applies to both `B1.P3` and `B1.P3 2`.
- Create one domain that applies to one composite region, the latter referring to `B1.P3` and `B1.P3 2`.

For demonstration purposes, you will create a composite region and use it as the location for a single fluid domain.

Create a single region that includes both pipe-flange regions:

1. Create a new composite region by selecting **Insert** > **Regions** > **Composite Region**.
2. In the **Insert Region** dialog box, set the name to `CatConverter`.
3. Click **OK**.
4. Apply the following settings:

Tab	Setting	Value
Basic Settings	Dimension (Filter)	3D
	Region List	B1.P3, B1.P3 2

5. Click **OK**.

12.6.2. Creating the Fluid Domain

For this simulation you will use a thermal energy heat transfer model and assume turbulent flow. Create the fluid domain using the composite region that you created earlier:

1. Ensure that no default domain is present under `Flow Analysis 1`. If a default domain is present, right-click it and select **Delete**.
2. Create a new domain by selecting **Insert** > **Domain**, or click *Domain*  .
3. In the **Insert Domain** dialog box, set the name to `Pipes`.
4. Click **OK**.
5. Apply the following settings (the values are taken from the problem description):

Tab	Setting	Value
Basic Settings	Location and Type > Location	CatConverter
	Location and Type > Domain Type	Fluid Domain
	Fluid and Particle Definitions	Fluid 1
	Fluid and Particle Definitions > Fluid 1 > Material	Air Ideal Gas
	Domain Models > Pressure > Reference Pressure	1 [atm]
Fluid Models	Heat Transfer > Option	Thermal Energy

6. Click **OK**.

12.6.3. Creating the Porous Domain

The catalyst-coated honeycomb structure will be modeled using a porous domain with a directional source of quadratic resistance, as described in the problem description. The streamwise directional resistance is aligned with the Z axis.

For quadratic resistances, the pressure drop is modeled using:

$$\frac{\partial p}{\partial x_i} = -K_Q \, |\boldsymbol{U}| \, U_i \qquad \textbf{(12–1)}$$

where K_Q is the quadratic resistance coefficient, U_i is the local velocity in the i direction, and $\frac{\partial p}{\partial x_i}$ is the pressure drop gradient in the i direction.

1. Create a new domain by selecting **Insert** > **Domain**, or click *Domain*  .
2. In the **Insert Domain** dialog box, set the name to `Housing`.
3. Click **OK**.
4. Apply the following settings:

Tab	Setting	Value
Basic Settings	Location and Type > Location	LIVE [1 (p. 199)]
	Location and Type > Domain Type	Porous Domain
	Fluid and Particle Definitions	Fluid 1
	Fluid and Particle Definitions > Fluid 1 > Material	Air Ideal Gas
	Solid Definitions	(Add a new solid named `Steel`)
	Solid Definitions > Steel > Material	Steel
	Domain Models > Pressure > Reference Pressure	1 [atm]
Fluid Models	Heat Transfer > Option	Thermal Energy
Solid Models	Heat Transfer > Option	Thermal Energy
Porosity Settings	Volume Porosity > Option	Value
	Volume Porosity > Volume Porosity	Porosity [4 (p. 199)]
	Loss Model > Option	Directional Loss
	Loss Model > Loss Velocity Type	Superficial [2 (p. 199)]
	Loss Model > Directional Loss > Streamwise Direction > Option	Cartesian Components
	Loss Model > Directional Loss > Streamwise Direction > X Component	0
	Loss Model > Directional Loss > Streamwise Direction > Y Component	0
	Loss Model > Directional Loss > Streamwise Direction > Z Component	-1
	Loss Model > Directional Loss > Streamwise Loss > Option	Linear and Quadratic Resistance Coefficients
	Loss Model > Directional Loss > Streamwise Loss > Quadratic Resistance Coefficient	(Selected)
	Loss Model > Directional Loss > Streamwise Loss > Quadratic Resistance Coefficient > Quadratic Coefficient	650 [kg m^-4] [3 (p. 199)]
	Loss Model > Directional Loss > Transverse Loss > Option	Streamwise Coefficient Multiplier
	Loss Model > Directional Loss > Transverse Loss > Multiplier	10 [3 (p. 199)]
	Fluid Solid Area Density > Interfacial Area Den.	AreaDen [4 (p. 199)]
	Fluid Solid Heat Transfer > Heat Trans. Coeff.	HTC [4 (p. 199)]

Footnotes

1. This is the entire housing section as predefined in the mesh.
2. Superficial velocity is the velocity at which the flow would travel if the porosity of the domain were 100%. It is less than the true velocity.
3. From the problem description.
4. In order to enter an expression, you must first click *Enter Expression*  .

5. Click **OK**.

12.6.4. Creating and Editing the Boundaries

Create the inlet and outlet boundaries using the values given in the problem description.

12.6.4.1. Creating the Inlet Boundary

1. Create a new boundary in domain `Pipes` named `Inlet`.
2. Apply the following settings:

Tab	Setting	Value
Basic Settings	Boundary Type	Inlet
	Location	PipeEnd 2
Boundary Details [1 (p. 199)]	Mass and Momentum > Normal Speed	25 [m s^-1] [2 (p. 199)]
	Heat Transfer > Static Temperature	Tinlet [3 (p. 199)]

Footnotes

1. The default level of turbulence is suitable for this application.
2. From the problem description.
3. In order to enter an expression, you must first click *Enter Expression*  .

3. Click **OK**.

12.6.4.2. Creating the Outlet Boundary

Set up the outlet with a static pressure boundary:

1. Create a new boundary in domain `Pipes` named `Outlet`.
2. Apply the following settings:

Tab	Setting	Value
Basic Settings	Boundary Type	Outlet

Tab	Setting	Value
	Location	PipeEnd
Boundary Details	Mass and Momentum > Option	Static Pressure
	Mass and Momentum > Relative Pressure	0 [Pa]

3. Click **OK**.

   The remaining outer surfaces are automatically assigned to the default (no slip wall) boundaries: `Housing Default` and `Pipes Default`.

12.6.4.3. Editing the Housing Default Boundary

In order to model the heat transfer through the `Housing` domain, several parameters from the `Housing Default` boundary need to be modified.

1. In the **Outline** tree, right-click `Housing Default` and select **Edit**.
2. Apply the following settings:

Tab	Setting	Value
Boundary Details	Heat Transfer > Option	Heat Transfer Coefficient
	Heat Transfer > Heat Trans. Coeff.	HTCoutside [1 (p. 200)]
	Heat Transfer > Outside Temperature	Toutside [1 (p. 200)]
Solid Values	Boundary Conditions > Steel > Heat Transfer Option	Heat Transfer Coefficient
	Boundary Conditions > Steel > Heat Transfer > Heat Trans. Coeff.	HTCoutside [1 (p. 200)]
	Boundary Conditions > Steel > Heat Transfer > Outside Temperature	Toutside [1 (p. 200)]

Footnote

1. In order to enter an expression, you must first click *Enter Expression* 

3. Click **OK**.

12.6.5. Creating the Domain Interfaces

You will next create a pair of domain interfaces to model the connection between the fluid and porous domains. The meshes on the interfaces are dissimilar, so a *General Grid Interface* (GGI) connection method is required. Domain interfaces are capable of modeling changes in reference frame as well as other changes that are not applicable to this simulation.

Two interfaces are required, one to connect the inlet flange to the catalyst housing and one to connect the outlet flange to the catalyst housing.

1. Create a new domain interface by selecting **Insert** > **Domain Interface**, or click *Domain Interface*  .
2. In the **Insert Domain Interface** dialog box, set the name to `InletSide`.
3. Click **OK**.
4. Apply the following settings:

Tab	Setting	Value
Basic Settings	Interface Type	Fluid Porous
	Interface Side 1 > Domain (Filter)	Pipes
	Interface Side 1 > Region List	FlangeEnd 2
	Interface Side 2 > Domain (Filter)	Housing
	Interface Side 2 > Region List	INLET
	Mesh Connection Method > Mesh Connection > Option	GGI

5. Click **OK**.
6. Create a similar domain interface named `OutletSide` that connects `FlangeEnd` (in domain `Pipes`) to `OUTLET` (in domain `Housing`).

12.6.6. Setting Initial Values

A sensible guess for the initial velocity is the expected velocity through the catalyst housing. You can assume incompressible flow and apply conservation of mass to obtain an approximate velocity of 2.8 [m s^-1] through the housing based on the following known information:

- The inlet velocity: 25 [m s^-1]
- The cross sectional area of the inlet and housing, which can be determined using the function calculator in CFD-Post: 0.001913 m^2 and 0.024039 m^2 respectively
- The porosity of the honeycomb structure: 70%

1. Click *Global Initialization*  .
2. Apply the following settings:

Tab	Setting	Value
Global Settings	Initial Conditions > Cartesian Velocity Components > Option	Automatic with Value
	Initial Conditions > Cartesian Velocity Components > U	0 [m s^-1]
	Initial Conditions > Cartesian Velocity Components > V	0 [m s^-1]

Tab	Setting	Value
	Initial Conditions > Cartesian Velocity Components > W	-2.8 [m s^-1]

3. Click **OK**.

12.6.7. Setting Solver Control

Assuming velocities of 25 m/s in the inlet and outlet pipes, and 2.8 m/s in the catalyst housing, an approximate fluid residence time of 0.1 s can be calculated. A sensible time step is 1/4 to 1/2 of the fluid residence time. In this case, use a time step of 0.04 s.

For the convergence criteria, an RMS value of at least 1e-05 is usually required for adequate convergence, but the default value of 1e-04 is sufficient for demonstration purposes.

1. Click *Solver Control*  .
2. Apply the following settings:

Tab	Setting	Value
Basic Settings	Convergence Control > Fluid Timescale Control > Timescale Control	Physical Timescale
	Convergence Control >Fluid Timescale Control > Physical Timescale	0.04 [s]

3. Click **OK**.

12.6.8. Setting a Discretization Option

The `porous cs discretisation option` expert parameter specifies how the pressure is treated at interfaces to a porous domain: 1=constant static pressure; 2=constant total pressure. Constant total pressure is the preferred and more physical setting. However, when using this setting, in cases where there are sections of the porous interface where there is little or no flow normal to the interface, the CFX-Solver may fail to converge. These convergence difficulties may be overcome by using the less physical constant static pressure setting.

This simulation involves flow that moves from a fluid domain into a porous domain, approaching the interface at various angles. In this case, better convergence can be achieved by changing the `porous cs discretisation option` expert parameter from the default value of 2 to 1.

1. Click **Insert** > **Solver** > **Expert Parameter**.
2. Apply the following settings in the **Expert Parameters** details view:

Tab	Setting	Value
Discretization	Miscellaneous > porous cs discretization option	(Selected)
	Miscellaneous > porous cs discretization option > Value	1

3. Click **OK**.

12.6.9. Writing the CFX-Solver Input (.def) File

1. Click *Define Run*  .
2. Apply the following settings:

Setting	Value
File name	CatConv.def

3. Click **Save**.

   CFX-Solver Manager automatically starts and, on the **Define Run** dialog box, the **Solver Input File** is set.
4. If using Standalone Mode, quit CFX-Pre, saving the simulation (.cfx) file at your discretion.

12.7. Obtaining a Solution Using CFX-Solver Manager

At this point, CFX-Solver Manager is running.

1. Ensure that the **Define Run** dialog box is displayed.
2. Click **Start Run**.

   CFX-Solver runs and attempts to obtain a solution. This may take a long time, depending on your system. Eventually a dialog box is displayed.
3. Select the check box next to **Post-Process Results** when the completion message appears at the end of the run.
4. If using Standalone Mode, select the check box next to **Shut down CFX-Solver Manager**.
5. Click **OK**.

12.8. Viewing the Results in CFD-Post

In this section, you will inspect the GGI interface to see the lack of node alignment that characterizes this type of interface. You will also create plots showing the distribution of temperature and pressure on a flat plane that intersects the catalytic converter. You will then make a chart showing pressure versus Z coordinate using data from a polyline that runs around the perimeter of the catalytic converter. Finally, you will export data from the polyline to a file. Such a file could be used in other programs, or could be loaded into CFD-Post (for example, to use as data for a chart line).

The topics in this section include:

12.8.1. Viewing the Mesh on a GGI Interface

In this section, you will examine a GGI interface. As a preliminary step, do the following:

1. When CFD-Post opens, if you see the Domain Selector dialog box, ensure that both domains are selected, then click **OK**.

2. Edit `User Locations and Plots` > `Wireframe`.
3. Set **Edge Angle** to `10 [degree]` and click `Apply` to see more of the mesh surface.
4. Turn off the visibility of `User Locations and Plots` > `Wireframe`.
5. Right-click a blank area in the viewer and select **Predefined Camera** > **View From -Z**.

In the **Outline** tree view, four interface sides are listed. There are two sides to the interface between the housing and the inlet. Similarly, there are two sides to the corresponding interface on the outlet side.

Examine the interface on the inlet side to see the nature of the GGI connection:

1. In the **Outline** tree view, edit `InletSide Side 1`.
2. Apply the following settings:

Tab	Setting	Value
Render	Show Faces	(Cleared)
	Show Mesh Lines	(Selected)
	Show Mesh Lines > Color Mode	User Specified
	Show Mesh Lines > Line Color	(Red)

3. Click **Apply**.
4. In the **Outline** tree view, edit `InletSide Side 2`.
5. Apply the following settings:

Tab	Setting	Value
Render	Show Faces	(Cleared)
	Show Mesh Lines	(Selected)
	Show Mesh Lines > Color Mode	User Specified
	Show Mesh Lines > Line Color	(Green)

6. Click **Apply**.
7. Click *Fit View*  to re-center and re-scale the geometry.

You can now see the tetrahedral/prism and hexahedral mesh on each side of the GGI interface. This interface was used to produce a connection between dissimilar meshes before the solution was calculated. Notice that there are more tetrahedral/prism elements than hexahedral elements and that the extent of the two meshes is not quite the same (this is most noticeable on the curved edges).

12.8.2. Creating User Locations

In this section, you will create locators that you will use to make plots.

To make it easier to see the locators, adjust the view as follows:

1. Turn on the visibility of `User Locations and Plots` > `Wireframe`.
2. Turn off the visibility of `Pipes` > `InletSide Side 1` and `Housing` > `InletSide Side 2`.

12.8.2.1. Creating a Slice Plane

Later in this tutorial, you will produce a contour plot and a vector plot to observe pressure changes. Both of these plots require a slice plane locator.

Create a slice plane through the geometry as follows:

1. Right-click a blank area in the viewer and select **Predefined Camera** > **View From +Y**.
2. Create a new plane named `Plane 1`.
3. Apply the following settings:

Tab	Setting	Value
Geometry	Definition > Method	ZX Plane
	Definition > Y	0.0 [m]
Color	Mode	Variable
	Variable	Steel.Temperature[1 (p. 205)]
	Range	Global

Footnote

1. If particular items do not appear in the drop-down list, click the *Ellipsis* icon ... to see all available items.

4. Click **Apply**. Notice the temperature distribution in the steel throughout the catalytic converter housing.
5. To see the temperature distribution in the fluid, change **Variable** to **Temperature** and click **Apply**.
6. Turn off the visibility of `User Locations and Plots` > `Plane 1` after you have analyzed the air temperature variation on `Plane 1`.

12.8.2.2. Creating a User Surface

You will create a user surface to observe the different characteristics of steel and air at the catalytic converter wall. Because the materials have different properties, the characteristics of each will vary slightly.

1. Click on **Location**  and select **User Surface**.
2. Click **OK** to accept the default name.
3. Apply the following settings:

Tab	Setting	Value
Geometry	Method	Transformed Surface
	Surface Name	Housing Default
Color	Mode	Variable
	Variable	Steel.Temperature[1 (p. 206)]
	Range	Local

Footnote

1. If particular items do not appear in the drop-down list, click the *Ellipsis* icon ... to see all available items.

4. Click **Apply**. Observe that the steel temperature decreases as it reaches the outlet pipe.
5. Apply the following changes to show the air temperature at the wall:

Tab	Setting	Value
Color	Variable	Temperature
	Range	Local

6. Click **Apply**.
7. Apply the following settings to show the steel heat flux at the wall:

Tab	Setting	Value
Color	Variable	Steel.Wall Heat Flux[1 (p. 206)]
	Range	Local[2 (p. 206)]

Footnote

1. If particular items do not appear in the drop-down list, click the *Ellipsis* icon ... to see all available items.
2. The values of the heat flux are negative because heat flow is directed away from the catalyst housing.

8. Click **Apply**. The heat flux should be greatest where the inlet pipe meets the housing body.
9. Apply the following settings to show the air heat flux at the wall:

Tab	Setting	Value
Color	Variable	Wall Heat Flux[1 (p. 206)]
	Range	Local[2 (p. 206)]

Footnote

1. If particular items do not appear in the drop-down list, click the *Ellipsis* icon ... to see all available items.
2. The values of the heat flux are negative because heat flow is directed away from the catalytic converter.

10. Click **Apply**.
11. Turn off the visibility of `User Locations and Plots` > `User Surface 1`.

12.8.2.3. Creating a Polyline

Later in this tutorial, you will make a chart that plots data from a polyline.

The **Method** used to create the polyline can be `From File`, `Boundary Intersection`, or `From Contour`. If you select `From File`, you must specify a file containing point definitions in the required format.

In this tutorial, you will use the `Boundary Intersection` method. This creates a polyline from the intersecting line between a boundary object and a location (for example, between a wall and a plane). The points on the polyline are where the intersecting line cuts through a surface mesh edge.

You will be able to see the polyline following the intersecting line between the wall, inlet and outlet boundaries and the slice plane.

1. Create a new polyline named `Polyline 1`.
2. Apply the following settings:

Tab	Setting	Value
Geometry	Method	Boundary Intersection
	Boundary List	Housing Default, Pipes Default [1 (p. 207)]
	Intersect With	Plane 1
Color	Mode	Constant
	Color	(Yellow)
Render	Line Width	3

Footnote

1. Click the *Ellipsis* icon ... to select multiple items using the **Ctrl** key.

3. Click **Apply**.
4. Turn off the visibility of `User Locations and Plots` > `Polyline 1`.

12.8.3. Creating Plots

In this section, you will make plots on the slice plane and polyline locators.

12.8.3.1. Creating a Contour Plot of Pressure

You will now create a contour plot to observe the pressure change throughout the main body of the catalytic converter:

1. Clear `Plane 1` in the **Outline** tab if you have not.already done so.

2. Create a new contour plot named `Contour 1`.
3. Apply the following settings:

Tab	Setting	Value
Geometry	Locations	Plane 1
	Variable	Pressure
	Range	Global
	# of Contours	30 [1 (p. 208)]
Render	Show Contour Bands	(Cleared)

Footnote

1. Determined by experiment.

4. Click **Apply**.

   From the contour plot, you can see that the pressure falls steadily through the main body of the catalytic converter.

12.8.3.2. Creating a Vector Plot on the Slice Plane

Create a vector plot to display the recirculation zone:

1. Create a new vector plot named `Vector 1`.
2. Apply the following settings:

Tab	Setting	Value
Geometry	Locations	Plane 1
Symbol	Symbol Size	0.1
	Normalize Symbols	(Selected)

3. Click **Apply**.

Notice that the flow separates from the walls, where the inlet pipe expands into the flange, setting up a recirculation zone. The flow is uniform through the catalyst housing.

12.8.3.3. Creating a Chart of Pressure versus the Z Coordinate

In this section, you will make a chart to see if the pressure drop is, as expected, linear by plotting a line graph of pressure against the z-coordinate. In this case you will use CFD-Post to produce the graph, but you could also export the data, then read it into any standard plotting package.

1. Create a new chart named `Chart 1`.
2. Apply the following settings:

Tab	Setting	Value
General	Title	Pressure Drop through a Catalytic Converter
Data Series	Name	Pressure Drop
	Data Source > Location	Polyline 1
X Axis	Data Selection > Variable	Z
Y Axis	Data Selection > Variable	Pressure
Line Display	Line Display > Line Style	None
	Line Display > Symbols	Rectangle
Chart Display	Sizes > Symbol	3

3. Click **Apply**.

   Through the main body of the catalytic converter you can see that the pressure drop is linear. This is in the region from approximately Z=0.06 to Z=0.26. The two lines show the pressure on each side of the wall. You can see a noticeable difference in pressure between the two walls on the inlet side of the housing (at around Z=0.26).

4. If required, in the **Outline** tree view, select `Contour 1` and `Vector 1`.
5. Click the **3D Viewer** tab, then right-click a blank area and select **Predefined Camera** > **View From +Y**.

   You should now see that the flow enters the housing from the inlet pipe at a slight angle, producing a higher pressure on the high X wall of the housing.

12.8.4. Exporting Polyline Data

You can export data from a polyline for use in other software.

Export data as follows:

1. From the main menu, select **File** > **Export** > **Export**.
2. Apply the following settings:

Tab	Setting	Value
Options	Locations	Polyline 1
	Export Geometry Information	(Selected) [1 (p. 209)]
	Select Variables	Pressure
Formatting	Precision	3

Footnote

1. This ensures X, Y, and Z to be sent to the output file.

3. Click **Save**.

The file `export.csv` will be written to the current working directory in a comma-separated variable format. This file can be opened in any text editor. You can use the exported data file to plot charts in other software such as a Microsoft Excel spreadsheet.

4. When finished, quit CFD-Post.

Chapter 13: Non-Newtonian Fluid Flow in an Annulus

This tutorial includes:

13.1. Tutorial Features

In this tutorial you will learn about:

- Defining a non-Newtonian fluid.
- Using the Moving Wall feature to apply a rotation to the fluid at a wall boundary.

Component	Feature	Details
CFX-Pre	User Mode	General Mode
	Analysis Type	Steady State
	Fluid Type	General Fluid
	Domain Type	Single Domain
	Turbulence Model	Laminar
	Heat Transfer	None
	Boundary Conditions	Symmetry Plane
		Wall: No-Slip
		Wall: Moving
	Timestep	Auto Time Scale
CFD-Post	Plots	Sampling Plane
		Vector

13.2. Background Theory

A Newtonian fluid is a fluid for which shear stress is linearly proportional to shear-strain rate, with temperature held constant. For such a fluid, the dynamic viscosity is constant and equal to the shear stress divided by the shear-strain rate.

A non-Newtonian fluid is a fluid for which the shear stress in not linearly proportional to the shear-strain rate. For such fluids, the *apparent viscosity* is the ratio of shear stress to shear-strain rate for a given shear-strain rate.

A shear-thickening fluid is a type of non-Newtonian fluid for which the apparent viscosity increases with increasing shear-strain rate.

Figure 13.1 Shear Stress of a Shear-thickening Fluid

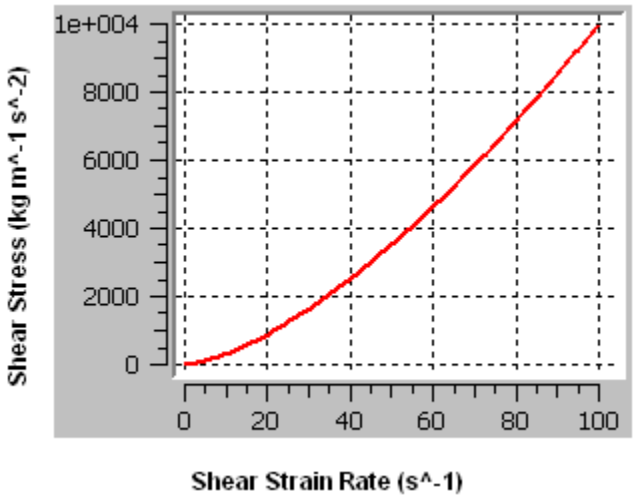
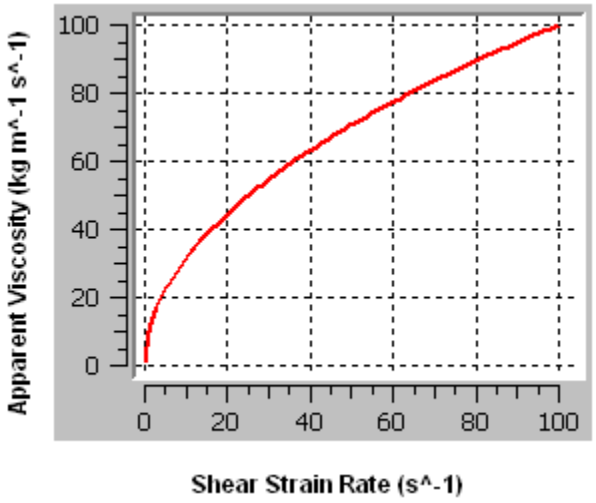


Figure 13.2 Apparent Viscosity of a Shear-thickening Fluid



This tutorial involves a shear thickening fluid that obeys the Ostwald de Waele model between apparent viscosity and shear-strain rate:

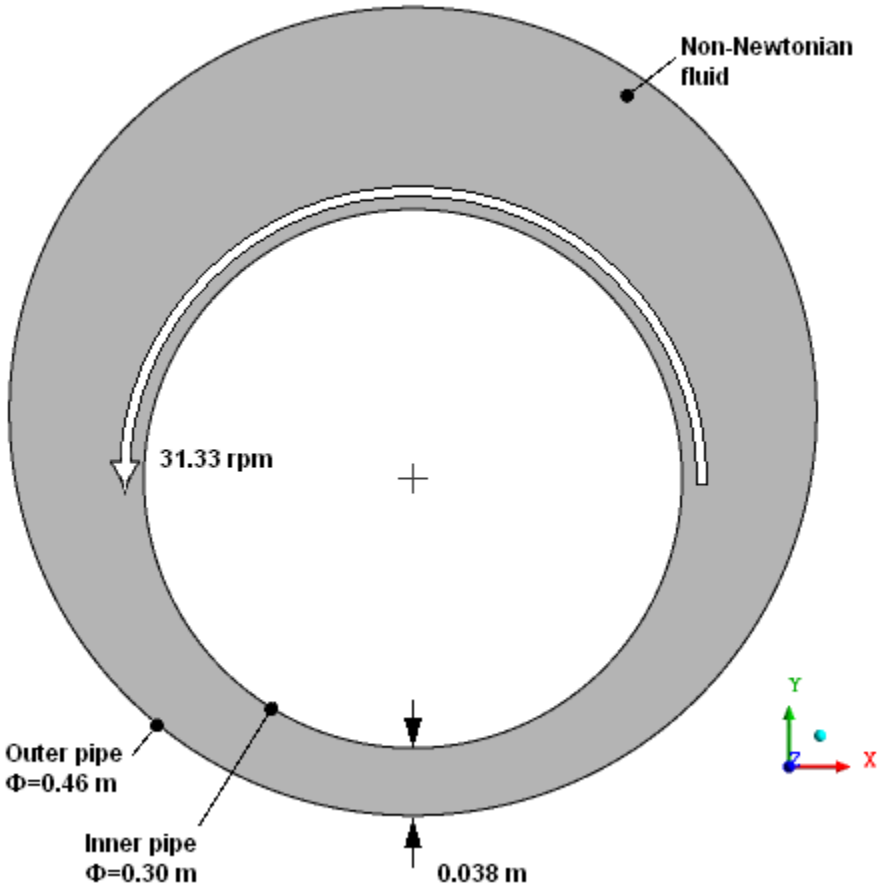
$$\mu = K\left(\lambda \dot{\gamma}\right)^{n-1} \qquad \textbf{(13–1)}$$

where μ is the apparent viscosity, K is the viscosity consistency, $\dot{\gamma}$ is the shear-strain rate, λ is a normalizing time constant, and n is the Power Law index. Note that the units for K are not tied to the value of n because the quantity in parentheses is dimensionless.

13.3. Overview of the Problem to Solve

In this tutorial, a shear-thickening liquid rotates in a 2D eccentric annular pipe gap. The outer pipe remains stationary while the inner pipe rotates at a constant rate about its own axis, which is the Z-axis. Both pipes have nonslip surfaces.

The fluid used in this simulation has material properties that are not a function of temperature. The ambient pressure is 1 atmosphere.



The shear-thickening liquid that is used in this tutorial obeys the Ostwald de Waele model with a viscosity consistency of 10.0 kg m^{-1} s^{-1}, a Power Law index of 1.5, and a time constant of 1 s. This model is assumed to be valid for shear-strain rates ranging from 1.0E-3 s^{-1} to 100 s^{-1}. The fluid has a density of 1.0E4 kg m^{-3}. The viscosity is plotted over this range in *Figure 13.2* (p. 212).

13.4. Before You Begin

If this is the first tutorial you are working with, it is important to review the following topics before beginning:

- *Preparing a Working Directory and Starting ANSYS CFX in Standalone Mode* (p. 1)
- *Running ANSYS CFX Tutorials in ANSYS Workbench* (p. 2)
- *Changing the Display Colors* (p. 5)
- *Playing a Tutorial Session File* (p. 4)

13.5. Starting CFX-Pre

1. Prepare the working directory using the following files in the `examples` directory:
   - `NonNewton.pre`
   - `NonNewtonMesh.gtm`
2. Set the working directory and start CFX-Pre.

   For details, see *Preparing a Working Directory and Starting ANSYS CFX in Standalone Mode* (p. 1)

13.6. Defining a Case in CFX-Pre

If you want to set up the simulation automatically using a tutorial session file, run `NonNewton.pre`. For details, see *Playing a Tutorial Session File* (p. 4). Then proceed to *Obtaining a Solution Using CFX-Solver Manager* (p. 218).

If you want to set up the simulation manually, proceed to the following steps:

1. In CFX-Pre, select **File** > **New Case**.
2. Select **General** and click **OK**.
3. Select **File** > **Save Case As**.
4. Under **File name**, type `NonNewton`.
5. Click **Save**.

13.6.1. Importing the Mesh

1. Edit `Case Options` > `General` in the **Outline** tree view and ensure that **Automatic Default Domain** is turned off.

   Default Domain generation should be turned off because you will create a new domain manually, later in this tutorial.
2. Right-click `Mesh` and select **Import Mesh** > **CFX Mesh**.
3. Apply the following settings:

Setting	Value
File name	NonNewtonMesh.gtm

4. Click **Open**.

13.6.2. Creating the Fluid

As stated in the problem description, the shear-thickening liquid that is used in this tutorial obeys the Ostwald de Waele model with a viscosity consistency (K) of 10.0 kg m^{-1} s^{-1}, a Power Law index (n) of 1.5, and a time constant of 1 s. This model is assumed to be valid for shear-strain rates ranging from 1.0E-3 s^{-1} to 100 s^{-1}. The fluid has a density of 1.0E4 kg m^{-3}.

1. Create a new material named `myfluid`.
2. Apply the following settings:

Tab	Setting	Value
Basic Settings	Thermodynamic State	(Selected)
	Thermodynamic State > Thermodynamic State	Liquid
Material Properties	Thermodynamic Properties > Equation of State > Molar Mass	1.0 [kg kmol^-1][a]
	Thermodynamic Properties > Equation of State > Density	1.0E+4 [kg m^-3]
	Transport Properties > Dynamic Viscosity	(Selected)
	Transport Properties > Dynamic Viscosity > Option	Non Newtonian Model

[a]This is not the correct value for molar mass, but this property is not used by CFX-Solver in this case. In other cases it might be used.

3. Apply the following settings under **Transport Properties** > **Dynamic Viscosity** > **Non Newtonian Viscosity Model**:

Setting	Value
Option	Ostwald de Waele
Viscosity Consistency	10.0 [kg m^-1 s^–1]
Min Shear Strain Rate	0.001 [s^-1]
Max Shear Strain Rate	100 [s^-1]
Time Constant	1 [s]
Power Law Index	1.5

4. Click **OK**.

13.6.3. Creating the Domain

The flow is expected to be laminar because the Reynolds number, based on the rotational speed, the maximum width of the pipe gap, and a representative viscosity (calculated using the shear-strain rate in the widest part of the gap, assuming a linear velocity profile), is approximately 30, which is well within the laminar-flow range.

From the problem description, the ambient pressure is 1 atmosphere.

Create a fluid domain that uses the non-Newtonian fluid you created in the previous section, and specify laminar flow with a reference pressure of 1 atmosphere:

1. Ensure that `Flow Analysis 1` > `Default Domain` is deleted. If not, right-click `Default Domain` and select **Delete**.
2. Click *Domain*  and set the name to `NonNewton`.
3. Apply the following settings to `NonNewton`:

Tab	Setting	Value
Basic Settings	Location	B8
	Fluid and Particle Definitions	Fluid 1
	Fluid and Particle Definitions > Fluid 1 > Material	myfluid
Fluid Models	Heat Transfer > Option	None
	Turbulence > Option	None (Laminar)

4. Click **OK**.

13.6.4. Creating the Boundaries

The inner and outer pipes both have nonslip surfaces. A rotating-wall boundary is required for the inner pipe. For the outer pipe, which is stationary, the default boundary is suitable. By not explicitly creating a boundary for the outer pipe, the latter receives the default wall boundary.

This tutorial models 2D flow in a pipe gap, where the latter is infinite in the Z-direction. The flow domain models a thin 3D slice (in fact, just one layer of mesh elements) that has two surfaces of constant-Z coordinate that each require a boundary. Symmetry boundary conditions are suitable in this case, since there is no pressure gradient or velocity gradient in the Z-direction.

13.6.4.1. Wall Boundary for the Inner Pipe

From the problem description, the inner pipe rotates at 31.33 rpm about the Z-axis. Create a wall boundary for the inner pipe that indicates this rotation:

1. Create a new boundary named `rotwall`.
2. Apply the following settings:

Tab	Setting	Value
Basic Settings	Boundary Type	Wall
	Location	rotwall
Boundary Details	Mass And Momentum > Option	No Slip Wall
	Mass And Momentum > Wall Velocity	(Selected)
	Mass And Momentum > Wall Velocity > Option	Rotating Wall
	Mass And Momentum > Wall Velocity > Angular Velocity	31.33 [rev min^-1]
	Mass And Momentum > Axis Definition > Option	Coordinate Axis
	Mass And Momentum > Axis Definition > Rotation Axis	Global Z

3. Click **OK**.

13.6.4.2. Symmetry Plane Boundary

In order to simulate the presence of an infinite number of identical 2D slices while ensuring that the flow remains 2D, apply a symmetry boundary on the high-Z and low-Z sides of the domain:

1. Create a new boundary named SymP1.
2. Apply the following settings:

Tab	Setting	Value
Basic Settings	Boundary Type	Symmetry
	Location	SymP1

3. Click **OK**.
4. Create a new boundary named SymP2.
5. Apply the following settings:

Tab	Setting	Value
Basic Settings	Boundary Type	Symmetry
	Location	SymP2

6. Click **OK**.

   The outer annulus surfaces will default to the no-slip stationary wall boundary.

13.6.5. Setting Initial Values

A reasonable guess for the initial velocity field is a value of zero throughout the domain. In this case, the problem converges adequately and quickly with such an initial guess. If this were not the case, you could, in principle, create and use CEL expressions to specify a better approximation of the steady-state flow field based on the information given in the problem description.

Set a static initial velocity field:

1. Click *Global Initialization*  .
2. Apply the following settings:

Tab	Setting	Value
Global Settings	Initial Conditions > Cartesian Velocity Components > Option	Automatic with Value
	Initial Conditions > Cartesian Velocity Components > U	0 [m s^-1]
	Initial Conditions > Cartesian Velocity Components > V	0 [m s^-1]
	Initial Conditions > Cartesian Velocity Components > W	0 [m s^-1]

3. Click **OK**.

13.6.6. Setting Solver Control

Because this flow is low-speed, laminar, and because of the nature of the geometry, the solution converges very well. For this reason, set the solver control settings for a high degree of accuracy and a high degree of convergence.

1. Click *Solver Control*  .
2. Apply the following settings:

Tab	Setting	Value
Basic Settings	Advection Scheme > Option	Specified Blend Factor
	Advection Scheme > Blend Factor	1.0[a]
	Convergence Control > Max. Iterations	50
	Convergence Criteria > Residual Type	RMS
	Convergence Criteria > Residual Target	1e-05[b]

[a]This is the most accurate but least robust advection scheme.

[b]This target demands a solution with a very high degree of convergence. For more information about recommended convergence targets, see Judging Convergence.

3. Click **OK**.

13.6.7. Writing the CFX-Solver Input (.def) File

1. Click *Define Run*  .
2. Apply the following settings:

Setting	Value
File name	NonNewton.def

3. Click **Save**.

   CFX-Solver Manager automatically starts and, on the **Define Run** dialog box, the **Solver Input File** is set.
4. If using Standalone Mode, quit CFX-Pre, saving the simulation (`.cfx`) file at your discretion.

13.7. Obtaining a Solution Using CFX-Solver Manager

When CFX-Pre has shut down and CFX-Solver Manager has started, you can obtain a solution to the CFD problem by following the instructions below:

1. Ensure that the **Define Run** dialog box is displayed.
2. Click **Start Run**.

   CFX-Solver runs and attempts to obtain a solution. This may take a long time, depending on your system. Eventually a dialog box is displayed.

3. Select the check box next to **Post-Process Results** when the completion message appears at the end of the run.
4. If using Standalone Mode, select the check box next to **Shut down CFX-Solver Manager**.
5. Click **OK**.

13.8. Viewing the Results in CFD-Post

The following steps instruct you on how to create a vector plot showing the velocity values in the domain.

1. Right-click a blank area in the viewer and select **Predefined Camera** > **View From -Z** from the shortcut menu.
2. Create a new plane named `Plane 1`.

   This plane will be used as a locator for a vector plot. To produce regularly-spaced sample points, create a circular sample plane, centered on the inner pipe, with a radius sufficient to cover the entire domain, and specify a reasonable number of sample points in the radial and theta directions. Note that the sample points are generated over the entire plane, and only those that are in the domain are usable in a vector plot.
3. Apply the following settings:

Tab	Setting	Value
Geometry	Definition > Method	Point and Normal
	Definition > Point	0, 0, 0.015[a]
	Definition > Normal	0, 0, 1
	Plane Bounds > Type	Circular
	Plane Bounds > Radius	0.3 [m]
	Plane Type	Sample
	Plane Type > R Samples	32
	Plane Type > Theta Samples	24
Render	Show Faces	(Cleared)
	Show Mesh Lines	(Selected)
	Show Mesh Lines > Color Mode	User Specified
	Line Color	(Choose green, or some other color, to distinguish the sample plane from the `Wireframe` object.)

[a]This is the point on the axis of the inner pipe, in the middle of the domain in the Z-direction.

4. Click **Apply**.
5. Examine the sample plane. The sample points are located at the line intersections. Note that many of the sample points are outside the domain. Only those points that are in the domain are usable for positioning vectors in a vector plot.
6. Turn off the visibility of `Plane 1`.
7. Create a new vector plot named `Vector 1` on `Plane 1`.

8. Apply the following settings:

Tab	Setting	Value
Geometry	Definition > Locations	Plane 1
	Definition > Sampling	Vertex[a]
	Definition > Reduction	Reduction Factor
	Definition > Factor	1.0[b]
	Definition > Variable	Velocity
	Definition > Boundary Data	Hybrid[c]
Symbol	Symbol Size	3[d]

[a]This causes the vectors to be located at the nodes of the sample plane you created previously. Note that the vectors can alternatively be spaced using other options that do not require a sample plane. For details, see Sampling.

[b]A reduction factor of 1.0 causes no reduction in the number of vectors so that there will be one vector per sample point.

[c]The hybrid values are modified at the boundaries for post-processing purposes. For details, see Hybrid and Conservative Variable Values.

[d]Because CFD-Post normalizes the size of the vectors based on the largest vector, and because of the large variation of velocity in this case, the smallest velocity vectors would normally be too small to see clearly.

9. Click **Apply**.

In CFX-Pre, you created a shear-thickening liquid that obeys the Ostwald de Waele model for shear-strain rates ranging from 1.0E-3 s^{-1} to 100 s^{-1}. The values of dynamic viscosity, which are a function of the shear-strain rate, were calculated as part of the solution. You can post-process the solution using these values, which are stored in the `Dynamic Viscosity` variable. For example, you can use this variable to color graphics objects.

Color `Plane 1` using the `Dynamic Viscosity` variable:

1. Turn on the visibility of `Plane 1`.
2. Edit `Plane 1`.
3. Apply the following settings:

Tab	Setting	Value
Color	Mode	Variable
	Variable	Dynamic Viscosity
Render	Show Faces	(Selected)

4. Click **Apply**

Try plotting `Shear Strain Rate` on the same plane. Note that the distribution is somewhat different than that of `Dynamic Viscosity`, as a consequence of the non-linear relationship (see *Figure 13.2* (p. 212)).

When you have finished, quit CFD-Post.

Chapter 14: Flow in an Axial Rotor/Stator

This tutorial includes:

14.1. Tutorial Features

In this tutorial you will learn about:

- Using the Turbo Wizard in CFX-Pre to quickly specify a turbomachinery application.
- Multiple Frames of Reference and Generalized Grid Interface.
- Using a Frozen Rotor interface between the rotor and stator domains.
- Modifying an existing simulation.
- Setting up a transient calculation.
- Using a Transient Rotor-Stator interface condition to replace a Frozen Rotor interface.
- Creating a transient animation showing domain movement in CFD-Post.

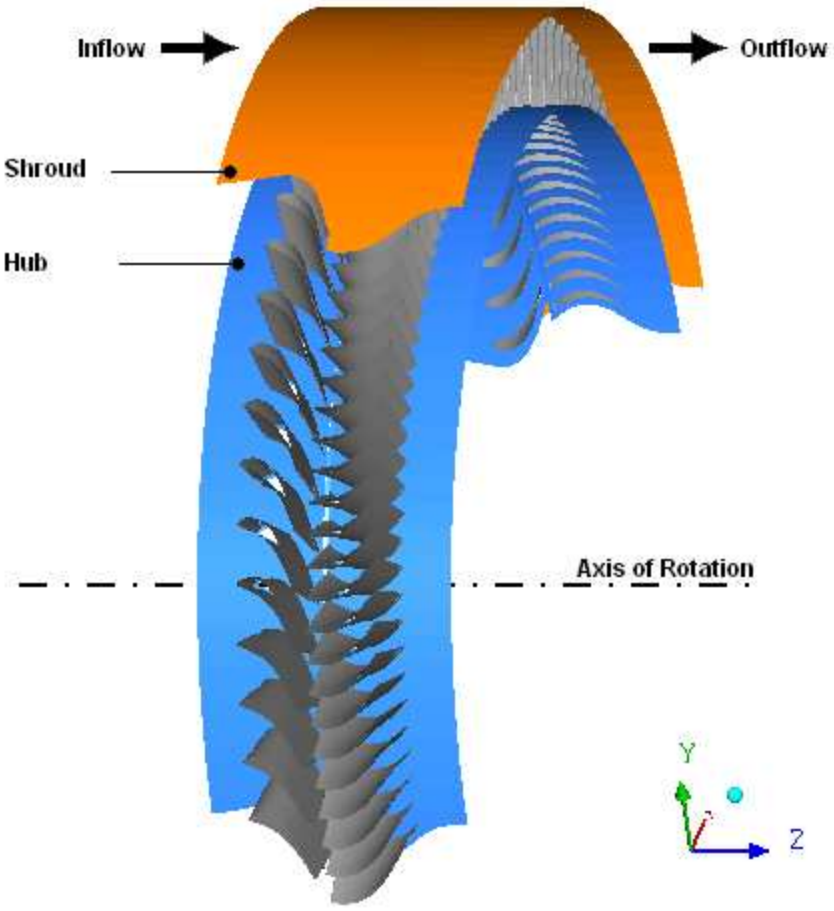
Component	Feature	Details
CFX-Pre	User Mode	Turbo Wizard
	Analysis Type	Steady State
		Transient
	Fluid Type	Ideal Gas
	Domain Type	Multiple Domain
		Rotating Frame of Reference
	Turbulence Model	k-Epsilon
	Heat Transfer	Total Energy
	Boundary Conditions	Inlet (Subsonic)
		Outlet (Subsonic)
		Wall: No-Slip
		Wall: Adiabatic
	Domain Interfaces	Frozen Rotor

Component	Feature	Details
		Periodic
		Transient Rotor Stator
	Timestep	Physical Time Scale
		Transient Example
	Transient Results File	
CFX-Solver Manager	Restart	
	Parallel Processing	
CFD-Post	Plots	Animation
		Isosurface
		Surface Group
	Turbo Post	
	Other	Changing the Color Range
		Chart Creation
		Instancing Transformation
		Movie Generation
		Quantitative Calculation
		Time Step Selection
		Transient Animation

14.2. Overview of the Problem to Solve

The goal of this tutorial is to set up a transient calculation of the Rotor-Stator to then create an animation showing its domain movement.

The full geometry of the axial rotor/stator contains 60 stator blades and 113 rotor blades. The following figure shows approximately half of the full geometry. The Inflow and Outflow labels show the location of the modeled section in *Figure 14.1* (p. 224).



The geometry to be modeled consists of a single stator blade passage and two rotor blade passages. This is an approximation to the full geometry since the ratio of rotor blades to stator blades is close to, but not exactly, 2:1. In the stator blade passage a 6° section is being modeled (360°/60 blades), while in the rotor blade passage, a 6.372° section is being modeled (2*360°/113 blades). This produces a pitch ratio at the interface between the stator and rotor of 0.942. As the flow crosses the interface, it is scaled to allow this type of geometry to be modeled. This results in an approximation of the inflow to the rotor passage. Furthermore, the flow across the interface will not appear continuous due to the scaling applied.

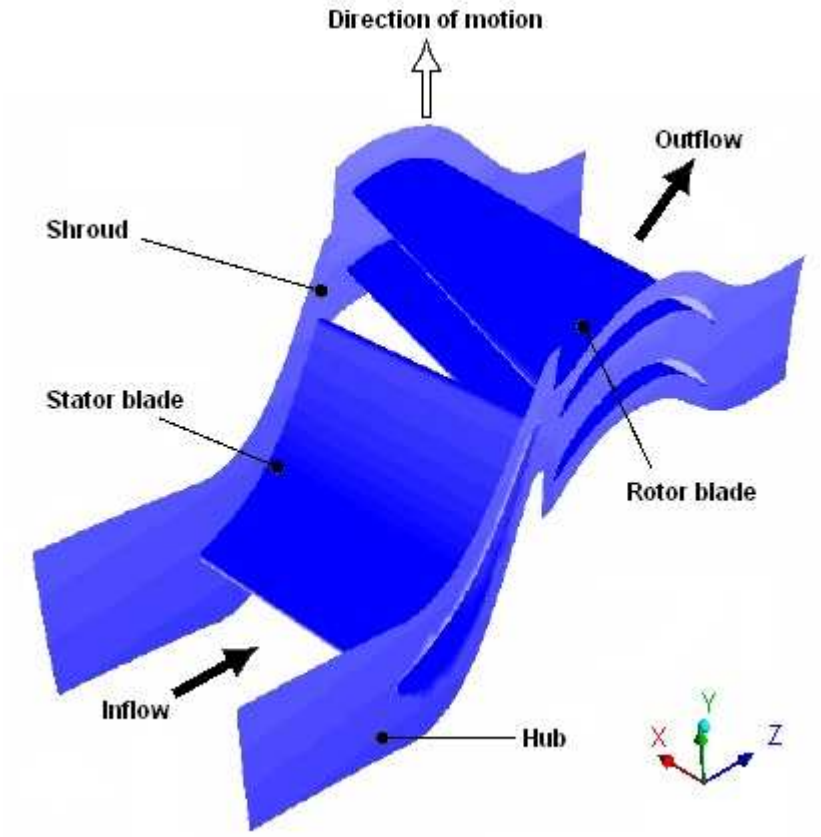
You should always try to obtain a pitch ratio as close to 1 as possible in your model to minimize approximations, but this must be weighed against computational resources. A full machine analysis can be performed (modeling all rotor and stator blades) which will always eliminate any pitch change, but will require significant computational time. For this rotor/stator geometry, a 1/4 machine section (28 rotor blades, 15 stator blades) would produce a pitch change of 1.009, but this would require a model about 15 times larger than in this tutorial example.

In this example, the rotor rotates about the Z-axis at 523.6 rad/s while the stator is stationary. Periodic boundaries are used to allow only a small section of the full geometry to be modeled.

The important parameters of this problem are:

- Total pressure = 0.265 bar
- Static Pressure = 0.0662 bar
- Total temperature = 328.5 K

Figure 14.1 Geometry subsection



The overall approach to solving this problem is to first define the Frozen Rotor simulation using the Turbomachinery wizard. The mesh for the rotor created in CFX-TASCflow will then be imported and combined with a second mesh (the stator), which was created using CFX-Mesh. The results will be viewed using the Turbo-Post feature. The existing Frozen Rotor simulation will then be modified to define the transient Rotor-Stator simulation. The transient Rotor-Stator simulation will be performed using the steady-state Frozen Rotor as an initial guess. Finally, a transient animation showing domain movement will be created in CFD-Post.

14.3. Before You Begin

If this is the first tutorial you are working with, it is important to review the following topics before beginning:

- *Preparing a Working Directory and Starting ANSYS CFX in Standalone Mode* (p. 1)
- *Running ANSYS CFX Tutorials in ANSYS Workbench* (p. 2)
- *Changing the Display Colors* (p. 5)
- *Playing a Tutorial Session File* (p. 4)

14.4. Starting CFX-Pre

1. Prepare the working directory using the following files in the `examples` directory:
   - `Axial.pre`
   - `AxialIni.pre`

- `AxialIni_001.res`
- `rotor.grd`
- `stator.gtm`

2. Set the working directory and start CFX-Pre.

   For details, see *Preparing a Working Directory and Starting ANSYS CFX in Standalone Mode* (p. 1)

14.5. Defining a Frozen Rotor Case in CFX-Pre

If you want to set up the simulation automatically using a tutorial session file, run `AxialIni.pre`. For details, see *Playing a Tutorial Session File* (p. 4). Then proceed to *Obtaining a Solution to the Frozen Rotor Model* (p. 228).

Otherwise, this simulation will be set up manually using the Turbomachinery wizard in CFX-Pre. This pre-processing mode is designed to simplify the setup of turbomachinery simulations.

1. In CFX-Pre, select **File** > **New Case**.
2. Select **Turbomachinery** and click **OK**.
3. Select **File** > **Save Case As**.
4. Under **File name**, type `AxialIni`.
5. Click **Save**.

14.5.1. Basic Settings

1. Set **Machine Type** to `Axial Turbine`.
2. Click **Next**.

14.5.2. Component Definition

Two new components are required. As they are created, meshes are imported.

1. Right-click in the blank area and select **Add Component** from the shortcut menu.
2. Create a new component of type `Stationary`, named S1.
3. Apply the following setting

Setting	Value
Mesh > File	stator.gtm[a]

[a]You may have to select the `CFX Mesh (*gtm *cfx)` option under **Files of type**.

4. Create a new component of type `Rotating`, named R1.
5. Apply the following settings

Setting	Value
Component Type > Value	523.6 [radian s^-1]
Mesh > File	rotor.grd[a]

Setting	Value
Options > Mesh Units	m

[a]You may have to select the `CFX-TASCflow (*grd)` option under **Files of Type**.

Note

The components must be ordered as above (stator then rotor) in order for the interface to be created correctly. The order of the two components can be changed by right-clicking on S1 and selecting **Move Component Up**.

When a component is defined, Turbo Mode will automatically select a list of regions that correspond to certain boundary types. This information should be reviewed in the **Region Information** section to ensure that all is correct. This information will be used to help set up boundary conditions and interfaces. The upper case turbo regions that are selected (e.g., HUB) correspond to the region names in the CFX-TASCflow grd file. CFX-TASCflow turbomachinery meshes use these names consistently.

6. Click **Passages and Alignment** > **Edit**.
7. Set **Passages and Alignment** > **Passages/Mesh** > **Passages per Mesh** to 2.
8. Ensure that **Passages and Alignment** > **Passages to Model** is set to 2.
9. Click **Passages and Alignment** > **Done**.
10. Click **Next**.

14.5.3. Physics Definition

In this section, you will set properties of the fluid domain and some solver parameters.

1. Apply the following settings

Tab	Setting	Value
Physics Definition	Fluid	Air Ideal Gas
	Analysis Type > Type	Steady State
	Model Data > Reference Pressure	0.25 [atm]
	Model Data > Heat Transfer	Total Energy
	Model Data > Turbulence	k-Epsilon
	Inflow/Outflow Boundary Templates > P-Total Inlet Mass Flow Outlet	(Selected)
	Inflow/Outflow Boundary Templates > Inflow > P-Total	0 [atm]
	Inflow/Outflow Boundary Templates > Inflow > T-Total	340 [K]
	Inflow/Outflow Boundary Templates > Inflow > Flow Direction	Normal to Boundary
	Inflow/Outflow Boundary Templates > Outflow > Mass Flow	Per Component
	Inflow/Outflow Boundary Templates > Outflow > Mass Flow Rate	0.06 [kg s^-1]

Tab	Setting	Value
	Interface > Default Type	Frozen Rotor
	Solver Parameters > Advection Scheme	High Resolution
	Solver Parameters > Convergence Control	Physical Times-cale
	Solver Parameters > Physical Timescale	0.002 [s][a]

[a]This time scale is approximately equal to 1 /ω, which is often appropriate for rotating machinery applications.

2. Click **Next**.

14.5.4. Interface Definition

CFX-Pre will try to create appropriate interfaces using the region names presented previously in the **Region Information** section. In this case, you should see that a periodic interface has been generated for both the rotor and the stator. These are required when modeling a small section of the true geometry. An interface is also required to connect the two components together across the frame change.

1. Review the various interfaces but do not change them.
2. Click **Next**.

14.5.5. Boundary Definition

CFX-Pre will try to create appropriate boundary conditions using the region names presented previously in the **Region Information** section. In this case, you should see a list of boundary conditions that have been generated. They can be edited or deleted in the same way as the interface connections that were set up earlier.

1. Review the various boundary definitions but do not change them.
2. Click **Next**.

14.5.6. Final Operations

1. Set **Operation** to `Enter General Mode`.
2. Click **Finish**.

   After you click **Finish**, a dialog box appears stating that the Turbo report will not be included in the solver file because you are entering General Mode.
3. Click **Yes** to continue.

14.5.7. Writing the CFX-Solver Input (.def) File

1. Click *Define Run*  .
2. Apply the following settings:

Setting	Value
File name	AxialIni.def

3. Click **Save**.

CFX-Solver Manager automatically starts and, on the **Define Run** dialog box, the **Solver Input File** is set.

4. If using Standalone Mode, quit CFX-Pre, saving the simulation (`.cfx`) file at your discretion.

14.6. Obtaining a Solution to the Frozen Rotor Model

Compared to previous tutorials, the mesh for this tutorial contains many more nodes (although it is still too coarse to perform a high quality CFD simulation). This results in a corresponding increase in solution time for the problem. Solving this problem in parallel is recommended, if possible. Your machine should have a minimum of 256MB of memory to run this tutorial.

More detailed information about setting up CFX to run in parallel is available. For details, see *Flow Around a Blunt Body* (p. 111).

You can solve this example using Serial, Local Parallel or Distributed Parallel.

- *Obtaining a Solution in Serial* (p. 228)
- *Obtaining a Solution With Local Parallel* (p. 228)
- *Obtaining a Solution with Distributed Parallel* (p. 229)

14.6.1. Obtaining a Solution in Serial

If you do not have a license to run CFX in parallel you can run in serial by clicking the **Start Run** button when CFX-Solver Manager has opened up. Solution time in serial is approximately 45 minutes on a 1GHz processor.

1. Click **Start Run** on the **Define Run** dialog box.

   CFX-Solver runs and attempts to obtain a solution. This may take a long time, depending on your system. Eventually a dialog box is displayed.
2. Select the check box next to **Post-Process Results** when the completion message appears at the end of the run.
3. If using Standalone Mode, select the check box next to **Shut down CFX-Solver Manager**.
4. Click **OK**.

When you are finished, proceed to *Viewing the Frozen Rotor Results in CFD-Post* (p. 229).

14.6.2. Obtaining a Solution With Local Parallel

To run in local parallel, the machine you are on must have more than one processor.

1. Set **Run Mode** to a parallel mode suitable for your environment; for example, `HP MPI Local Parallel`.
2. If required, click *Add Partition*  to add more partitions.

   By default, 2 partitions are assigned.
3. Click **Start Run**.
4. Select the check box next to **Post-Process Results** when the completion message appears at the end of the run.

5. If using Standalone Mode, select the check box next to **Shut down CFX-Solver Manager**.
6. Click **OK**.

When you are finished, proceed to *Viewing the Frozen Rotor Results in CFD-Post* (p. 229).

14.6.3. Obtaining a Solution with Distributed Parallel

1. Set **Run Mode** to a parallel mode suitable for your environment; for example, `HP MPI Distributed Parallel`.

   One partition should already be assigned to the host that you are logged into.

2. Click *Insert Host*  to specify a new parallel host.
3. In **Select Parallel Hosts**, select another host name (this should be a machine that you can log into using the same user name).
4. Click **Add**, and then **Close**.

   The names of the two selected machines should be listed in the **Host Name** column of the **Define Run** dialog box.

5. Click **Start Run**.
6. Select the check box next to **Post-Process Results** when the completion message appears at the end of the run.
7. If using Standalone Mode, select the check box next to **Shut down CFX-Solver Manager**.
8. Click **OK**.

14.7. Viewing the Frozen Rotor Results in CFD-Post

The Turbo-Post feature will be demonstrated in the following sections. This feature is designed to greatly reduce the effort taken to post-process turbomachinery simulations. For details, see "Turbo Workspace"

14.7.1. Initializing Turbo-Post

When initializing turbo components, span, a (axial), r (radial), and Theta coordinates are generated for each component. Therefore, after entering the Turbo workspace and initializing the turbo components, you will be ready to start using the turbo-specific features offered in the Turbo workspace immediately. These features include Turbo Tree View, Turbo Surface, Turbo Line and Turbo Plots. For details see "Turbo Workspace"

To initialize Turbo-Post, the properties of each component must be set up. This includes information about the inlet, outlet, hub, shroud, blade, and periodic regions. It also includes information about the number of instances of each turbo component needed to represent the full geometry around the rotation axis, and the number of blade passages in the mesh for each turbo component.

1. When CFD-Post starts, the **Domain Selector** dialog box might appear. If it does, ensure that both the R1 and S1 domains are selected, then click **OK** to load the results from these domains.
2. Click the **Turbo** tab.

   The **Turbo Initialization** dialog box is displayed, and asks you whether you want to auto-initialize all components.

Note

If you do not see the **Turbo Initialization** dialog box, or as an alternative to using that dialog box, you can initialize all components by clicking the **Initialize All Components** button which is visible initially by default, or after double-clicking the **Initialization** object in the **Turbo** tree view.

3. Click **Yes**.

   In this case, the initialization works without problems. If there was a problem initializing a component, this would likely be indicated in the tree view.

14.7.2. Viewing Three Domain Passages

Next, you will create an instancing transformation to plot three domain passages; three blade passages for the stator and six blade passages for the rotor. It was chosen to create three times the geometry that was used in the simulation to help visualize the variation of pressure. Seeing neighboring passages will give a better understanding of the pressure variation through the axial rotor/stator.

The instancing properties of each domain have already been entered during Initialization. In the next steps, you will create a surface group plot to color the blade and hub surfaces with the same variable.

1. From the main menu, select **Insert** > **Location** > **Surface Group**.
2. Click **OK**.

   The default name is accepted.
3. Apply the following settings

Tab	Setting	Value
Geometry	Locations	R1 Blade, R1 Hub, S1 Blade, S1 Hub
Color	Mode	Variable
	Variable	Pressure

4. Click **Apply**.
5. Click the **Turbo** tab.
6. Open **Plots** > **3D View** for editing.
7. Apply the following settings

Tab	Setting	Value
3D View	Instancing > Domain	R1
	Instancing > # of Copies	3

8. Click **Apply**.
9. Apply the following settings

Tab	Setting	Value
3D View	Instancing > Domain	S1
	Instancing > # of Copies	3

10. Click **Apply**.
11. Click the **Outline** tab.

14.7.3. Blade Loading Turbo Chart

In this section, you will create a plot of pressure around the stator blade at a given spanwise location.

- In the **Turbo** tree view, double-click `Blade Loading`.

  This profile of the pressure curve is typical for turbomachinery applications.

  When you are finished viewing the chart, quit CFD-Post.

14.8. Setting up a Transient Rotor-Stator Calculation

This section describes the step-by-step definition of the flow physics in CFX-Pre. The existing steady-state frozen-rotor simulation is modified to define the transient rotor-stator simulation. If you have not already completed the frozen-rotor simulation, please refer to *Defining a Frozen Rotor Case in CFX-Pre* (p. 225) before proceeding with the transient rotor-stator simulation.

14.8.1. Playing a Session File

If you want to set up the simulation automatically and continue to *Obtaining a Solution to the Transient Rotor-Stator Model* (p. 234), run `Axial.pre`.

Note

The session file creates a new simulation named `Axial.cfx` and will not modify the existing database. It also copies the required initial values file from the examples directory to the current working directory.

14.8.2. Opening the Existing Case

This step involves opening the original simulation and saving it to a different location.

1. If CFX-Pre is not already running, start it.
2. Open the results file named `AxialIni_001.res`.
3. Save the case as `Axial.cfx` in your working directory.

14.8.3. Modifying the Physics Definition

You need to modify the domain to define a transient simulation. You are going to run for a time interval such that the rotor blades pass through 1 pitch (6.372°) using 10 time steps. This is generally too few time steps to obtain high quality results, but is sufficient for tutorial purposes. The time step size is calculated as follows:

Note

$$\text{RotationalSpeed} = 523.6 \text{ rad/s}$$
$$\text{Rotor Pitch Modeled} = 2 \cdot (2\pi \text{ rad}/113) \approx 0.1112 \text{ rad}$$
$$\text{Time to pass through 1 pitch} \approx (0.1112 \text{ rad}) / (523.6 \text{ rad/s}) \approx 2.124 \text{ e}-4 \text{ s}$$

Since 10 time steps are used over this interval each time step should be 2.124e-5 s.

1. Select **Tools** > **Turbo Mode**.

   Basic Settings is displayed.
2. Click **Next**.

   Component Definition is displayed.
3. Click **Next**.

   Physics Definition is displayed.
4. Apply the following settings

Tab	Setting	Value
Physics Definition	Fluid	Air Ideal Gas
	Analysis Type > Type	Transient
	Analysis Type > Total Time	2.124e-4 [s][a]
	Analysis Type > Time Steps	2.124e-5 [s][b]
	Interface > Default Type	Transient Rotor Stator

[a]This gives 10 timesteps of 2.124e-5 s

[b]This timestep will be used until the total time is reached

Note

A transient rotor-stator calculation often runs through more than one pitch. In these cases, it may be useful to look at variable data averaged over the time interval required to complete 1 pitch. You can then compare data for each pitch rotation to see if a "steady state" has been achieved, or if the flow is still developing.

5. Click **Next**.

   A warning message is displayed.
6. Click **Yes**.

   Interface Definition is displayed.
7. Click **Next**.

   Boundary Definition is displayed.

8. Click **Next**.

   Final Operations is displayed.
9. Ensure that **Operation** is set to `Enter General Mode`.
10. Click **Finish**.

    A message box notifies you that a turbo report will not be included in the solver file.
11. Click **Yes** to continue.

    Initial values are required, but will be supplied later using a results file.

14.8.4. Setting Output Control

1. Click *Output Control*  .
2. Click the **Trn Results** tab.
3. In the **Transient Results** tree view, click *Add new item*  , set **Name** to `Transient Results 1`, and click **OK**.
4. Apply the following settings

Option	Selected Variables
Output Variables List[a]	Pressure, Velocity, Velocity in Stn Frame
Output Frequency > Option	Time Interval
Output Frequency > Time Interval	2.124e-5 [s]

[a]Use the **Ctrl** key to select more than one variable.

5. Click **OK**.

14.8.5. Modifying Execution Control

1. Click *Execution Control*  .
2. Apply the following settings

Tab	Setting	Value
Run Definition	Solver Input File	Axial.def[a]

[a]You do not need to set the path unless you are planning on saving the solver file somewhere other than the working directory.

3. Confirm that the rest of the execution control settings are set appropriately.
4. Click **OK**.

14.8.6. Writing the CFX-Solver Input (.def) File

1. Click *Define Run*  .

   A warning will appear, due to a lack of initial values.

Initial values are required, but will be supplied later using a results file.

2. Click **Yes**.
3. If using Standalone Mode, quit CFX-Pre, saving the simulation (`.cfx`) file at your discretion.

14.9. Obtaining a Solution to the Transient Rotor-Stator Model

When the CFX-Solver Manager has started you will need to specify an initial values file before starting the CFX-Solver.

14.9.1. Serial Solution

If you do not have a license, or do not want to run CFX in parallel, you can run it in serial. Solution time in serial is similar to the first part of this tutorial.

1. Select **Run Definition** > **Initial Values Specification**.
2. Under **Initial Values Specification** > **Initial Values**, select `Initial Values 1`.
3. Under **Initial Values Specification** > **Initial Values** > **Initial Values 1 Settings** > **File Name**, click *Browse*  .
4. Select `AxialIni_001.res` from your working directory.
5. Click **Open**.
6. Under **Initial Values Specification** > **Use Mesh From**, select `Solver Input File`.
7. Click **Start Run**.

   CFX-Solver runs and attempts to obtain a solution. This can take a long time depending on your system. Eventually a dialog box is displayed.
8. When CFX-Solver is finished, select the check box next to **Post-Process Results**.
9. If using Standalone Mode, select the check box next to **Shut down CFX-Solver Manager**.
10. Click **OK**. Continue this tutorial from *Monitoring the Run* (p. 234).

14.9.2. Parallel Solution

Follow the first 6 steps of the serial procedure above (in *Serial Solution* (p. 234)), then perform the local parallel or distributed parallel procedure from the first part of this tutorial (in *Obtaining a Solution to the Frozen Rotor Model* (p. 228)).

14.9.3. Monitoring the Run

During the solution, look for the additional information that is provided for transient rotor-stator runs. Each time the rotor is rotated to its next position, the number of degrees of rotation and the fraction of a pitch moved is given. You should see that after 10 timesteps the rotor has been moved through 1 pitch.

14.10. Viewing the Transient Rotor-Stator Results in CFD-Post

To examine the transient interaction between the rotor and stator, you are going to create a blade-to-blade animation of pressure. A turbo surface will be used as the basis for this plot.

14.10.1. Initializing Turbo-Post

All the pre-processing will be done during the initialization of the turbo components. Only a few steps will therefore be required to display a surface of constant span and to create a turbo surface later on in this tutorial.

1. Click the **Turbo** tab.

   The **Turbo Initialization** dialog box is displayed and asks you whether you want to auto-initialize all components.

   Note

   If you do not see the **Turbo Initialization** dialog box, or as an alternative to using that dialog box, you can initialize all components by clicking the **Initialize All Components** button which is visible initially by default, or after double-clicking the **Initialization** object in the **Turbo** tree view.

2. Click **Yes**.

   Both components (domains) are now being initialized based on the automatically selected turbo regions. When the process is complete, a green turbine icon appears next to each component entry in the list. Also, the viewer displays a green background mesh for each initialized component.

3. Double-click `Component 1 (S1)` and review the automatically-selected turbo regions and other data in the details view.
4. Double-click `Component 2 (R1)` and review the automatically-selected turbo regions and other data in the details view (including the **Passages/Com** setting on the **Instancing** tab, which should have a value of 2).

14.10.2. Displaying a Surface of Constant Span

- In the **Turbo** tree view, double-click `Blade-to-Blade`.

  A surface of constant span appears, colored by pressure. This object can be edited and then redisplayed using the details view.

14.10.3. Using Multiple Turbo Viewports

1. In the **Turbo** tree view, double-click **Initialization**.
2. Click **Three Views**.

   Left view is **3D View**, top right is **Blade-to-Blade** and bottom right is **Meridional** view.

3. Click **Single View**.

14.10.4. Creating a Turbo Surface at Mid-Span

1. Create a **Turbo Surface** by selecting **Insert** > **Location** > **Turbo Surface** from the drop down menu with a **Constant Span** and value of 0.5.
2. Under the **Color** tab select **Variable** and set it to **Pressure** with a user specified range of -10000 [Pa] to -7000 [Pa].

14.10.5. Setting up Instancing Transformations

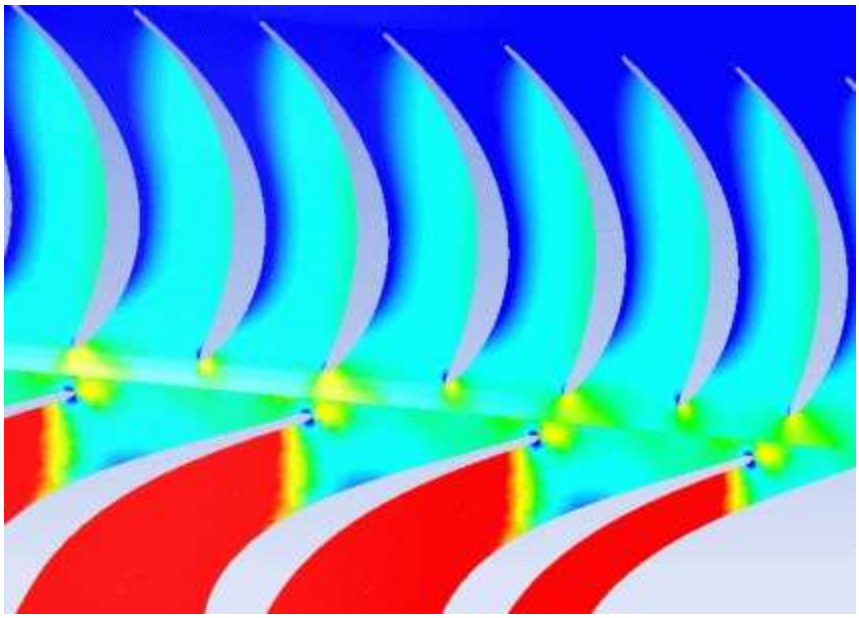
Next, you will use instancing transformations to view a larger section of the model. The properties for each domain have already been entered during the initialization phase, so only the number of instances needs to be set.

1. In the **Turbo** tree view, double-click the **3D View** object.
2. In the **Instancing** section of the form, set **# of Copies** to 6 for R1.
3. Click **Apply**.
4. In the **Instancing** section of the form, set **# of Copies** to 6 for S1.
5. Click **Apply**.
6. Return to the Outline tab and ensure that the turbo surface is visible again.

14.10.6. Animating the movement of the Rotor relative to the Stator

Start by loading the first timestep:

1. Click *Timestep Selector* .
2. Select time value 0.
3. Click **Apply** to load the timestep. The rotor blades move to their starting position. This is exactly 1 pitch from the previous position so the blades will not appear to move.
4. Turn off the visibility of `Wireframe`.
5. Position the geometry as shown below, ready for the animation. During the animation the rotor blades will move to the right. Make sure you have at least two rotor blades out of view to the left side of the viewer. They will come into view during the animation.



6. In the toolbar at the top of the window click *Animation* .
7. In the **Animation** dialog box, select the **Keyframe Animation** option.

8. Click *New*  to create `KeyFrameNo1`.
9. Select `KeyframeNo1`, then set **# of Frames** to 9, then press **Enter** while in the **# of Frames** box.

   Tip

   Be sure to press **Enter** and confirm that the new number appears in the list before continuing.

10. Use the **Timestep Selector** to load the final timestep.
11. In the **Animation** dialog box, click *New*  to create `KeyframeNo2`.
12. Click *More Animation Options*  to expand the **Animation** dialog box.
13. Click **Options** and set **Transient Case** to `TimeValue Interpolation`. Click **OK**.

    The animation now contains a total of 11 frames (9 intermediate frames plus the two Keyframes), one for each of the available time values.
14. In the expanded **Animation** dialog box, select **Save Movie**.
15. Set **Format** to `MPEG1`.
16. Click *Browse*  , next to the **Save Movie** box and then set the file name to an appropriate file name.
17. If frame 1 is not loaded (shown in the **F:** text box at the bottom of the **Animation** dialog box), click *To Beginning*  to load it.

    Wait for CFD-Post to finish loading the objects for this frame before proceeding.
18. Click *Play the animation*  .

    - It takes a while for the animation to complete.
    - To view the movie file, you will need to use a media player that supports the MPEG format.

    You will be able to see from the animation, and from the plots created previously, that the flow is not continuous across the interface. This is because a pitch change occurs. The relatively coarse mesh and the small number of timesteps used in the transient simulation also contribute to this. The movie was created with a narrow pressure range compared to the global range which exaggerates the differences across the interface.

14.10.7. Further Postprocessing

You can produce a report for the turbine as follows:

1. Click **File** > **Report** > **Report Templates**.
2. In the **Report Templates** dialog box, select **Turbine Report**, then click **Load**.

   The report will be generated automatically.
3. Click the **Report Viewer** tab (located below the viewer window).

   A report appears.

Note that a valid report depends on valid turbo initialization.

Chapter 15: Reacting Flow in a Mixing Tube

This tutorial includes:

15.1. Tutorial Features

In this tutorial you will learn about:

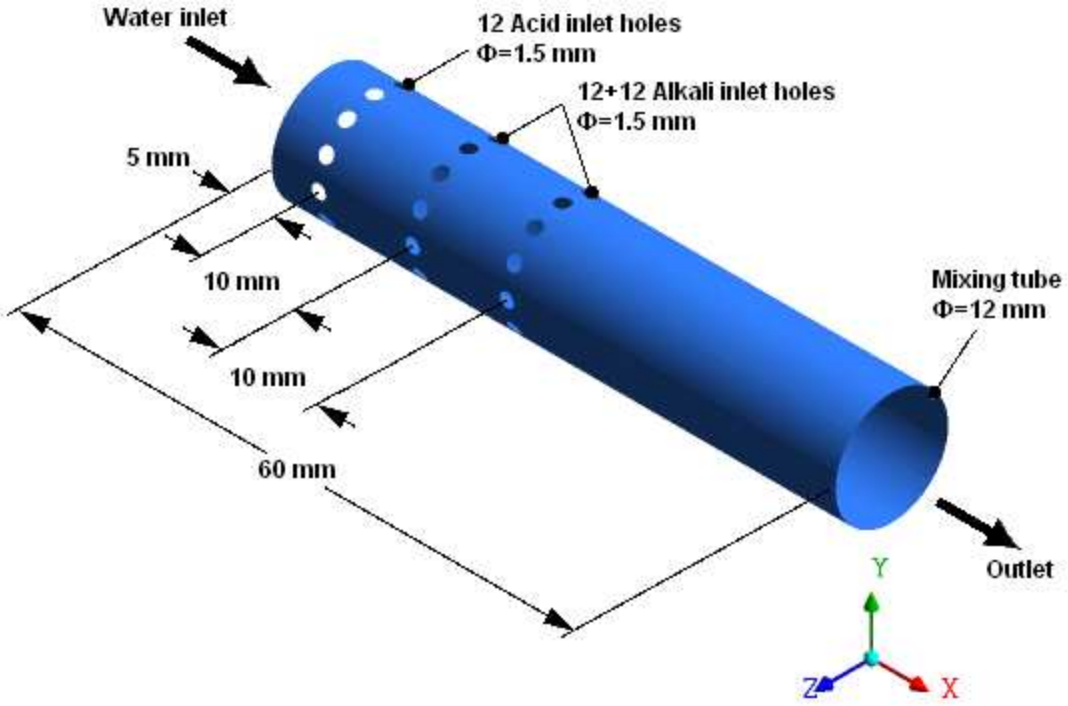
- Creating and using a multicomponent fluid in CFX-Pre.
- Using CEL to model a reaction in CFX-Pre.
- Using an algebraic Additional Variable to model a scalar distribution.
- Using a subdomain as the basis for component sources.

Component	Feature	Details
CFX-Pre	User Mode	General Mode
	Analysis Type	Steady State
	Fluid Type	Variable Composition Mixture
	Domain Type	Single Domain
	Turbulence Model	k-Epsilon
	Heat Transfer	Thermal Energy
	Particle Tracking	Component Source
	Boundary Conditions	Inlet (Subsonic)
		Outlet (Subsonic)
		Symmetry Plane
		Wall: Adiabatic
	Additional Variables	
	CEL (CFX Expression Language)	
	Timestep	Physical Time Scale
CFD-Post	Plots	Isosurface
		Slice Plane

15.2. Overview of the Problem to Solve

Reaction engineering is one of the core components in the chemical industry. Optimizing reactor design leads to higher yields, lower costs and, as a result, higher profit.

This example demonstrates the capability of ANSYS CFX to model basic reacting flows using a multicomponent fluid and CEL expressions.



The geometry consists of a mixing tube with three rings with twelve holes in each ring.

The main inlet has water entering at 2 m/s with a temperature of 300 K. The pressure at the outlet is 1 atm.

Through the ring of holes nearest the inlet, a solution of dilute sulfuric acid enters at 2 m/s with a temperature of 300 K. Through each of the two other rings of holes, a solution of dilute sodium hydroxide enters at 2.923 m/s with a temperature of 300 K. The properties of the solution of sulfuric acid are shown in *Table 15.1: Properties of the Dilute Sulfuric Acid Solution* (p. 240):

Table 15.1 Properties of the Dilute Sulfuric Acid Solution

Property	Value
Molar mass	19.517 kg kmol^-1
Density	1078 kg m^-3
Specific heat capacity	4190 J kg^-1 K^-1
Dynamic Viscosity	0.001 kg m^-1 s^-1
Thermal Conductivity	0.6 W m^-1 K^-1

Through the remaining two rings of holes, a solution of dilute sodium hydroxide (an alkali) enters with a temperature of 300K. The properties of the solution of sodium hydroxide are shown in *Table 15.2: Properties of the Dilute Sodium Hydroxide Solution* (p. 241).

Table 15.2 Properties of the Dilute Sodium Hydroxide Solution

Property	Value
Molar mass	18.292 kg kmol^-1
Density	1029 kg m^-3
Specific heat capacity	4190 J kg^-1 K^-1
Dynamic Viscosity	0.001 kg m^-1 s^-1
Thermal Conductivity	0.6 W m^-1 K^-1

The acid and alkali undergo an exothermic reaction to form a solution of sodium sulfate (a type of salt) and water according to the reaction:

$$H_2SO_4 + 2NaOH \rightarrow Na_2SO_4 + 2H_2O$$

Mixing the acid and alkali solutions in a stoichiometric ratio (and allowing them to react completely) would result in a salt water solution that would include water from each of the original solutions plus water produced during the reaction. The properties of this salt water product are shown in *Table 15.3: Properties of the Salt Water Product* (p. 241).

Table 15.3 Properties of the Salt Water Product

Setting	Value
Molar mass	18.600 kg kmol^-1
Density	1031 kg m^-3
Specific heat capacity	4190 J kg^-1 K^-1
Dynamic Viscosity	0.001 kg m^-1 s^-1
Thermal Conductivity	0.6 W m^-1 K^-1

The heat of reaction is 460 kJ per kg of dilute acid solution.

The flow is assumed to be fully turbulent and turbulence is assumed to have a significant effect on the reaction rate.

After running the simulation, you will plot the distribution of pH in the tube, and determine the extent to which the pH is neutralized at the outlet. You will also plot mass fraction distributions of acid, alkali and product.

15.3. Modeling Approach

In order to reduce memory requirements and solution time, only a 30° slice of the geometry will be modeled, and symmetry boundary conditions will be applied to represent the remaining geometry.

The reaction between acid and alkali is represented as a single-step irreversible liquid-phase reaction:

$$A + B \rightarrow C$$

Reagent A (dilute sulfuric acid) is injected through a ring of holes near the start of the tube. As it flows along the tube it reacts with Reagent B (dilute sodium hydroxide) which is injected through a further two rings of holes downstream. The product, C, remains in solution.

You will create a variable-composition mixture[1] that contains water, the reactants, and the product. To model the reaction, you will use CEL expressions to govern the mass sources for the acid, alkali and product components. You will also use CEL expressions to govern the thermal energy source. Providing mass and energy sources over a volume requires a subdomain. Because the reaction may occur anywhere in the domain, you will create a subdomain that occupies the entire flow domain.

To model the pH, you will create an algebraic Additional Variable that is governed by a CEL expression for pH. The Additional Variable will be available in the solution results for analysis during post-processing.

15.4. Before You Begin

If this is the first tutorial you are working with, it is important to review the following topics before beginning:

- *Preparing a Working Directory and Starting ANSYS CFX in Standalone Mode* (p. 1)
- *Running ANSYS CFX Tutorials in ANSYS Workbench* (p. 2)
- *Changing the Display Colors* (p. 5)
- *Playing a Tutorial Session File* (p. 4)

15.5. Starting CFX-Pre

1. Prepare the working directory using the following files in the `examples` directory:
   - `Reactor.pre`
   - `ReactorExpressions.ccl`
   - `ReactorMesh.gtm`
2. Set the working directory and start CFX-Pre.

   For details, see *Preparing a Working Directory and Starting ANSYS CFX in Standalone Mode* (p. 1)

15.6. Defining a Case in CFX-Pre

If you want to set up the simulation automatically using a tutorial session file, run `Reactor.pre`. For details, see *Playing a Tutorial Session File* (p. 4). Then proceed to *Obtaining a Solution Using CFX-Solver Manager* (p. 259).

If you want to set up the simulation manually, proceed to the following steps:

1. In CFX-Pre, select **File** > **New Case**.
2. Select **General** and click **OK**.
3. Select **File** > **Save Case As**.
4. Under **File name**, type `Reactor`.

[1] You can also model this type of reaction using a reacting mixture as your fluid. There is a tutorial that uses a reacting mixture: *Combustion and Radiation in a Can Combustor* (p. 343).

5. Click **Save**.

15.6.1. Importing the Mesh

1. Right-click `Mesh` and select **Import Mesh** > **CFX Mesh**.

   The **Import Mesh** dialog box appears.

2. Apply the following settings:

Setting	Value
File name	ReactorMesh.gtm

3. Click **Open**.

15.6.2. Creating a Multicomponent Fluid

In addition to providing template fluids, CFX allows you to create custom fluids for use in all your CFX models. A custom fluid may be defined as a pure substance, but may also be defined as a mixture, consisting of a number of transported fluid components. This type of fluid model is useful for applications involving mixtures, reactions, and combustion.

In order to define custom fluids, CFX-Pre provides the **Material** details view. This tool allows you to define your own fluids as pure substances, fixed composition mixtures or variable composition mixtures using a range of template property sets defined for common materials.

The mixing tube application requires a fluid made up from four separate materials (or components). The components are the reactants and products of a simple chemical reaction together with a neutral carrier liquid. You are first going to define the materials that take part in the reaction (acid, alkali and product) as pure substances. The neutral carrier liquid is water, and is already defined. Finally, you will create a variable composition mixture consisting of these four materials. This is the fluid that you will use in your simulation. A variable composition mixture (as opposed to a fixed composition mixture) is required because the proportion of each component will change throughout the simulation due to the reaction.

15.6.2.1. Acid Properties

The properties of the dilute sulfuric acid solution were stated in the problem description.

1. Create a new material named `acid`.
2. Apply the following settings:

Tab	Setting	Value
Basic Settings	Option	Pure Substance
	Thermodynamic State	(Selected)
	Thermodynamic State > Thermodynamic State	Liquid
Material Properties	Option	General Material
	Thermodynamic Properties > Equation of State > Option	Value

Tab	Setting	Value
	Thermodynamic Properties > Equation of State > Molar Mass	19.517 [kg kmol^-1][a]
	Thermodynamic Properties > Equation of State > Density	1078 [kg m^-3]
	Thermodynamic Properties > Specific Heat Capacity	(Selected)
	Thermodynamic Properties > Specific Heat Capacity > Option	Value
	Thermodynamic Properties > Specific Heat Capacity > Specific Heat Capacity	4190 [J kg^-1 K^-1]
	Transport Properties > Dynamic Viscosity	(Selected)
	Transport Properties > Dynamic Viscosity > Option	Value
	Transport Properties > Dynamic Viscosity > Dynamic Viscosity	0.001 [kg m^-1 s^-1]
	Transport Properties > Thermal Conductivity	(Selected)
	Transport Properties > Thermal Conductivity > Option	Value
	Transport Properties > Thermal Conductivity > Thermal Conductivity	0.6 [W m^-1 K^-1]

[a]The Molar Masses for the three materials do not affect the solution except through the post-processed variables Molar Concentration and Molar Fraction.

3. Click **OK**.

15.6.2.2. Alkali Properties

The properties of the dilute sodium hydroxide solution were stated in the problem description.

1. Create a new material named `alkali`.
2. Apply the following settings:

Tab	Setting	Value
Basic Settings	Option	Pure Substance
	Thermodynamic State	(Selected)
	Thermodynamic State > Thermodynamic State	Liquid
Material Properties	Option	General Material
	Thermodynamic Properties > Equation of State > Option	Value
	Thermodynamic Properties > Equation of State > Molar Mass	18.292 [kg kmol^-1]
	Thermodynamic Properties > Equation of State > Density	1029 [kg m^-3]

Tab	Setting	Value
	Thermodynamic Properties > Specific Heat Capacity	(Selected)
	Thermodynamic Properties > Specific Heat Capacity > Option	Value
	Thermodynamic Properties > Specific Heat Capacity > Specific Heat Capacity	4190 [J kg^-1 K^-1]
	Transport Properties > Dynamic Viscosity	(Selected)
	Transport Properties > Dynamic Viscosity > Option	Value
	Transport Properties > Dynamic Viscosity > Dynamic Viscosity	0.001 [kg m^-1 s^-1]
	Transport Properties > Thermal Conductivity	(Selected)
	Transport Properties > Thermal Conductivity > Option	Value
	Transport Properties > Thermal Conductivity > Thermal Conductivity	0.6 [W m^-1 K^-1]

3. Click **OK**.

15.6.2.3. Reaction Product Properties

The properties of the salt water product were stated in the problem description.

1. Create a new material named `product`.
2. Apply the following settings:

Tab	Setting	Value
Basic Settings	Option	Pure Substance
	Thermodynamic State	(Selected)
	Thermodynamic State > Thermodynamic State	Liquid
Material Properties	Option	General Material
	Thermodynamic Properties > Equation of State > Option	Value
	Thermodynamic Properties > Equation of State > Molar Mass	18.600 [kg kmol^-1]
	Thermodynamic Properties > Equation of State > Density	1031 [kg m^-3]
	Thermodynamic Properties > Specific Heat Capacity	(Selected)
	Thermodynamic Properties > Specific Heat Capacity > Option	Value

Tab	Setting	Value
	Thermodynamic Properties > Specific Heat Capacity > Specific Heat Capacity	4190 [J kg^-1 K^-1]
	Transport Properties > Dynamic Viscosity	(Selected)
	Transport Properties > Dynamic Viscosity > Option	Value
	Transport Properties > Dynamic Viscosity > Dynamic Viscosity	0.001 [kg m^-1 s^-1]
	Transport Properties > Thermal Conductivity	(Selected)
	Transport Properties > Thermal Conductivity > Option	Value
	Transport Properties > Thermal Conductivity > Thermal Conductivity	0.6 [W m^-1 K^-1]

3. Click **OK**.

15.6.2.4. Fluid Properties

Define a variable composition mixture by combining water with the three materials you have defined: `acid`, `alkali`, `product`.

1. Create a new material named `mixture`.
2. Apply the following settings:

Tab	Setting	Value
Basic Settings	Option	Variable Composition Mixture
	Material Group	User, Water Data
	Materials List	Water, acid, alkali, product
	Thermodynamic State	(Selected)
	Thermodynamic State > Thermodynamic State	Liquid

3. Click **OK**.

15.6.3. Creating an Additional Variable to Model pH

You are going to use an Additional Variable to model the distribution of pH in the mixing tube. You can create Additional Variables and use them in selected fluids in your domain.

1. Create a new Additional Variable named `MixturePH`.
2. Apply the following settings:

Tab	Setting	Value
Basic Settings	Variable Type	Specific

Tab	Setting	Value
	Units	[kg kg^-1]
	Tensor Type	Scalar

3. Click **OK**.

This Additional Variable is now available for use when you create or modify a domain. You will set other properties of the Additional Variable, including how it is calculated, when you apply it to the domain later in this tutorial.

15.6.4. Formulating the Reaction and pH as Expressions

This section includes:

- *Stoichiometric Ratio* (p. 247)
- *Reaction Source Terms* (p. 249)
- *Calculating pH* (p. 251)
- *Loading the Expressions to Model the Reaction and pH* (p. 252)

The first section shows a derivation for the mass-based stoichiometric ratio of alkali solution to acid solution. This ratio is used for calculating various quantities throughout this tutorial.

The second subsection (*Reaction Source Terms* (p. 249)) shows you how reactions and reaction kinetics can be formulated using the Eddy Break Up (EBU) model.

The third subsection (*Calculating pH* (p. 251)), shows you how pH is calculated.

In the fourth subsection (*Loading the Expressions to Model the Reaction and pH* (p. 252)) you will use a provided file to load CEL expressions for the reaction source terms and the pH.

15.6.4.1. Stoichiometric Ratio

The mass-based stoichiometric ratio of alkali solution to acid solution is a quantity that is used in several calculations in this tutorial. It represents the mass ratio of alkali solution to acid solution which leads to complete reaction with no excess alkali or acid (that is, neutral pH). This section of the tutorial shows you how to calculate the stoichiometric ratio, and introduces other quantities that are used in this tutorial.

The alkali solution contains water and sodium hydroxide. In the alkali solution, it is assumed that the sodium hydroxide molecules completely dissociate into ions according to the following reaction:

$$\mathrm{NaOH} \Rightarrow \mathrm{Na}^{+} + \mathrm{OH}^{-}$$

The acid solution contains water and sulfuric acid. In the acid solution, it is assumed that the sulfuric acid molecules completely dissociate into ions according to the following reaction:

$$\mathrm{H_2SO_4} \Rightarrow 2\mathrm{H}^{+} + \mathrm{SO_4}^{2-}$$

The Na^{+} ions and $\mathrm{SO_4}^{2-}$ ions react to form sodium sulfate (a type of salt) and water according to the reaction:

$$2\mathrm{Na}^{+} + 2\mathrm{OH}^{-} + 2\mathrm{H}^{+} + \mathrm{SO_4}^{2-} \Rightarrow \mathrm{Na_2SO_4} + 2\mathrm{H_2O}$$

Note that this reaction requires the ions from two molecules of sodium hydroxide and the ions from one molecule of sulfuric acid. The stoichiometric ratio for the dry alkali and acid molecules is 2-to-1.

Instead of modeling dry molecules of alkali and acid, this tutorial models solutions that contain these molecules (in dissociated form) plus water. The calculations used to model the alkali-acid reactions, and to measure the pH, require a mass-based stoichiometric ratio, i, that expresses the mass ratio between the alkali solution and the acid solution required for complete reaction of all of the (dissociated) alkali and acid molecules within them.

Using X^{2-} to denote $SO_4{}^{2-}$ and Y^{+} to denote Na^{+}, the ratio i can be computed as the ratio of the following two masses:

- The mass of alkali solution required to contain 2 kmol of Y^{+}
- The mass of acid solution required to contain 1 kmol of X^{2-}

A formula for calculating i is:

$$i = \frac{\left(\frac{2 \text{ kmol } Y^{+}}{\beta \text{ kmol } Y^{+}/\text{kg}}\right)}{\left(\frac{1 \text{ kmol } X^{2-}}{\alpha \text{ kmol } X^{2-}/\text{kg}}\right)} = \frac{2\alpha}{\beta} \qquad \textbf{(15–1)}$$

where:

- β is the concentration of $NaOH$ in kmol/kg solution (equal to the concentration of Y^{+} in kmol/kg solution).
- α is the concentration of H_2SO_4 in kmol/kg solution (equal to the concentration of X^{2-} in kmol/kg solution).

The molar mass of the alkali solution (given as 18.292 kg/kmol solution) is a weighted average of the molar masses of water (18.015 kg/kmol) and dry sodium hydroxide (39.9971 kg/kmol), with the weighting in proportion to the number of each type of molecule in the solution. You can compute the fraction of the molecules in the solution that are sodium hydroxide as:

$$\text{NaOH fraction} = \frac{\text{molar mass alkalisolution} - \text{molar mass } H_2O}{\text{molar mass NaOH} - \text{molar mass } H_2O}$$

$$= \frac{18.292 \text{ kg/kmol} - 18.015 \text{ kg/kmol}}{39.9971 \text{ kg/kmol} - 18.015 \text{ kg/kmol}}$$

$$= 0.012601 \text{ kmol NaOH/kmol solution}$$

β can then be calculated as follows:

$$\beta = \frac{0.012601 \text{ kmolNaOH/kmol solution}}{18.292 \text{ kg solution/kmol solution}} = 0.689\text{E-}3 \text{ kmol NaOH/kg solution}$$

The molar mass of the acid solution (given as 19.517 kg/kmol solution) is a weighted average of the molar masses of water (18.015 kg/kmol) and dry sulfuric acid (98.07848 kg/kmol), with the weighting in proportion to the number of each type of molecule in the solution. You can compute the fraction of the molecules in the solution that are sulfuric acid as:

$$\begin{aligned} H_2SO_4 \ \text{fraction} &= \frac{\text{molar mass acid solution} - \text{molar mass } H_2O}{\text{molar mass } H_2SO_4 - \text{molar mass } H_2O} \\ &= \frac{19.517 \ \text{kg/kmol} - 18.015 \ \text{kg/kmol}}{98.07848 \ \text{kg/kmol} - 18.015 \ \text{kg/kmol}} \\ &= 0.01876 \ \text{kmol } H_2SO_4/\text{kmol solution} \end{aligned}$$

α can then be calculated as follows:

$$\alpha = \frac{0.01876 \ \text{kmol } H_2SO_4/\text{kmol solution}}{19.517 \ \text{kg solution/kmolsolution}} = 0.961\text{E-}3 \ \text{kmol } H_2SO_4/\text{kg solution}$$

Substituting the values for β and α into *Equation 15–1* (p. 248) yields the mass-based stoichiometric ratio of alkali solution to acid solution: $i = 2.79$.

15.6.4.2. Reaction Source Terms

The reaction and reaction rate are modeled using a basic Eddy Break Up formulation for the component and energy sources. For example, the transport equation for the mass fraction of acid solution is:

$$\begin{aligned} d\,\frac{\partial}{\partial t}\left(\rho\, mf_{\text{acid}}\right) + \nabla\cdot\left(\rho\, U\ mf_{\text{acid}}\right) - \nabla\cdot\left(\rho\, D_{\text{A}}\ \nabla mf_{\text{acid}}\right) \\ = -4\,\rho\,\frac{\varepsilon}{k}\,min\left(mf_{\text{acid}}, \frac{mf_{\text{alkali}}}{i}\right) \end{aligned} \qquad \textbf{(15–2)}$$

where t is time, U is velocity, ρ is the local density of the variable composition mixture, mf_{acid} is the mass fraction of the acid solution in the mixture, D_{A} is the kinematic diffusivity of the acid solution through the mixture, and i is the stoichiometric ratio of alkali solution to acid solution based on mass fraction. The right-hand side represents the mass source term that is applied to the transport equation for the acid solution. The left-hand side consists of the transient, advection and diffusion terms.

In addition to specifying the sources for the acid solution and alkali solution, source coefficients will also be used in order to enhance solution convergence. For details, see the technical note at the end of this section.

The reaction rate is computed as:

$$\text{Reaction Rate} = \frac{4\rho\epsilon}{k}$$

where k is the turbulence kinetic energy, and ϵ is the turbulence eddy dissipation. Note that the reaction rate appears on the right-hand side of *Equation 15–2* (p. 249). The reaction rate is also used to govern the rate of thermal energy production according to the relation:

$$\text{Heat Source} = \text{Heat of Reaction} \times 4\,\rho\,\frac{\varepsilon}{k}\,min\left(mf_{\text{acid}}, \frac{mf_{\text{alkali}}}{i}\right)$$

From the problem description, the heat of reaction is 460 kJ per kg of acid solution.

Note

This is a technical note, for reference only.

A source is fully specified by an expression for its value S.

A source coefficient C is optional, but can be specified to provide convergence enhancement or stability for strongly-varying sources. The value of C may affect the rate of convergence but should not affect the converged results.

If no suitable value is available for C, the solution time scale or time step can still be reduced to help improve convergence of difficult source terms.

Important

C must never be positive.

An optimal value for C when solving an individual equation for a positive variable φ with a source S whose strength decreases with increasing φ is

$$C = \frac{\partial S}{\partial \varphi}$$

Where this derivative cannot be computed easily,

$$C = \frac{S}{\varphi}$$

may be sufficient to ensure convergence. (This is the form used for the acid solution and alkali solution mass source coefficients in this tutorial.)

Another useful formula for C is

$$C = -\frac{\rho}{\tau}$$

where τ is a local estimate for the source time scale. Provided that the source time scale is not excessively short compared to flow or mixing time scales, this may be a useful approach for controlling sources with positive feedback ($\partial S / \partial \varphi > 0$) or sources that do not depend directly on the solved variable φ.

15.6.4.3. Calculating pH

The pH (or acidity) of the mixture is a function of the volume-based concentration of H^+ ions. The latter can be computed using the following two equations, which are based on charge conservation and equilibrium conditions, respectively:

$$\left[\mathrm{H}^+\right]+\left[\mathrm{Y}^+\right]=2\left[\mathrm{X}^{2-}\right]+\left[\mathrm{OH}^-\right]$$

$$[\mathrm{H}^+]\ [\mathrm{OH}^-]=K_{\mathrm{W}}$$

(where K_{W} is the constant for the self-ionization of water (1.0E-14 kmol^2 m^{-6})).

You can substitute one equation into the other to obtain the following quadratic equation:

$$\left[\mathrm{H}^+\right]\left(\left[\mathrm{H}^+\right]+\left[\mathrm{Y}^+\right]-2\left[\mathrm{X}^{2-}\right]\right)=K_{\mathrm{W}}$$

which can be rearranged into standard quadratic form as:

$$[\mathrm{H}^+]^2+\left(\left[\mathrm{Y}^+\right]-2\left[\mathrm{X}^{2-}\right]\right)\left[\mathrm{H}^+\right]-K_{\mathrm{W}}=0$$

The quadratic equation can be solved for $[\mathrm{H}^+]$ using the equation $[H^+]=\frac{-b+\sqrt{\max\left[\left(b^2-4ac\right),0\right]}}{2a}$ where $a=1, b=\left[\mathrm{Y}^+\right]-2\left[\mathrm{X}^{2-}\right]$ and $c=-K_{\mathrm{W}}$.

The volume-based concentrations of X^{2-} and Y^+ are required to calculate b, and can be calculated from the mass fractions of the components using the following expressions:

$$\left[\mathrm{X}^{2-}\right]=\alpha\,\rho\left(mf_{\mathrm{acid}}+\frac{mf_{\mathrm{prod}}}{1+i}\right)$$

$$[\mathrm{Y}^+]=\beta\,\rho\left(mf_{\mathrm{alkali}}+\frac{i\,mf_{\mathrm{prod}}}{1+i}\right)$$

where:

- $[\mathrm{X}^{2-}]$ is the concentration of X^{2-} in kmol/m^3.
- $[\mathrm{Y}^+]$ is the concentration of Y^+ in kmol/m^3.
- α is the concentration of $\mathrm{H_2SO_4}$ in kmol/kg solution (equal to the concentration of X^{2-} in kmol/kg solution).

- β is the concentration of NaOH in kmol/kg solution (equal to the concentration of Y^+ in kmol/kg solution).
- ρ is the local density of the variable composition mixture.
- i is the mass-based stoichiometric ratio of alkali solution to acid solution.

Note that the second expression above can be re-written by substituting for β using *Equation 15–1* (p. 248).

The result is:

$$[\mathrm{Y}^+] = 2\alpha\ \rho\left(\frac{mf_{\mathrm{alkali}}}{i} + \frac{mf_{\mathrm{prod}}}{1+i}\right)$$

After solving for the concentration of H^+ ions, the pH can be computed as:

$$pH = -\log 10\ [H^+]$$

In order to set a limit on pH for calculation purposes, the following relation will be used in this tutorial:

$$pH = -\log 10\left(\max\left[\left[H^+\right], 10^{-15}\right]\right)$$

15.6.4.4. Loading the Expressions to Model the Reaction and pH

Load the expressions required to model the reaction sources and pH:

1. Select **File** > **Import** > **CCL**.
2. Ensure that **Import Method** is set to **Append**.
3. Select `ReactorExpressions.ccl`, which should be in your working directory.
4. Click **Open**.

Observe the expressions listed in the tree view of CFX-Pre. Some expressions are used to support other expressions. The main expressions are:

Expression Name	Description	Supporting Expressions
pH	The pH of the mixture.	Hions, a, b, c, Yions, Xions, alpha, i
HeatSource	The thermal energy released from the reaction.	HeatReaction, Rate
AcidSource	The rate of production of acid due to the reaction (always negative or zero).	Rate
AcidSourceCoeff	The source coefficient for AcidSource (to enhance convergence).	AcidSource

Expression Name	Description	Supporting Expressions
AlkaliSource	The rate of production of alkali due to the reaction (always negative or zero).	Rate
AlkaliSourceCoeff	The source coefficient for AlkaliSource (to enhance convergence).	AlkaliSource
ProductSource	The rate of production of salt water product (always positive or zero).	Rate

Note that the expressions do not refer to a particular fluid since there is only a single fluid (which happens to be a multicomponent fluid). In a multiphase simulation you must prefix variables with a fluid name, for example `Mixture.acid.mf` instead of `acid.mf`.

15.6.5. Creating the Domain

In this section, you will create a fluid domain that contains the variable composition mixture and the Additional Variable that you created earlier. The Additional Variable will be set up as an algebraic equation with values calculated from the CEL expression for pH.

1. Edit `Case Options` > `General` in the **Outline** tree view and ensure that **Automatic Default Domain** is turned on.

   A domain named `Default Domain` should now appear under the `Simulation` branch.

2. Double-click `Default Domain`. Under the **Fluid and Particle Definitions** setting, delete `Fluid 1` and create a new fluid definition called `Mixture`.

   Apply the following settings:

Tab	Setting	Value
Basic Settings	Location and Type > Location	B1.P3
	Location and Type > Domain Type	Fluid Domain
	Fluid and Particle Definitions	Mixture
	Fluid and Particle Definitions > Mixture > Material	mixture
	Domain Models > Pressure > Reference Pressure	1 [atm]
Fluid Models	Heat Transfer > Option	Thermal Energy
	Component Models > Component	acid
	Component Models > Component > acid > Option	Transport Equation
	Component Models > Component > acid > Kinematic Diffusivity	(Selected)
	Component Models > Component > acid > Kinematic Diffusivity > Kinematic Diffusivity	0.001 [m^2 s^-1]

3. Use the same **Option** and **Kinematic Diffusivity** settings for `alkali` and `product` as you have just set for `acid`.
4. For `Water`, set **Option** to `Constraint` as follows:

Tab	Setting	Value
Fluid Models	Component Models > Component	Water
	Component Models > Component > Water > Option	Constraint

One component must always use `Constraint`. This is the component used to balance the mass fraction equation; the sum of the mass fractions of all components of a fluid must equal unity.

5. Apply the following settings to apply the Additional Variable that you created earlier:

Tab	Setting	Value
Fluid Models	Additional Variable Models > Additional Variable > MixturePH	(Selected)
	Additional Variable Models > Additional Variable > MixturePH > Option	Algebraic Equation [a]
	Additional Variable Models > Additional Variable > MixturePH > Add. Var. Value	pH

[a]The other possible options either involve a transport equation to transport the Additional Variable in the flow field, or a Vector Algebraic Equation, which is for vector quantities. The Algebraic Equation is suitable because it allows the calculation of pH as a function of existing variables and expressions.

6. Click **OK**.

15.6.6. Creating a Subdomain to Model the Chemical Reactions

To provide the correct modeling for the chemical reaction you need to define mass fraction sources for the fluid components `acid`, `alkali`, and `product`. To do this, you need to create a subdomain where the relevant sources can be specified. In this case, sources need to be provided within the entire domain of the mixing tube since the reaction occurs throughout the domain.

1. Ensure that you have loaded the CEL expressions from the provided file.

   The expressions should be listed in the tree view.
2. Create a new subdomain named `sources`.
3. Apply the following settings:

Tab	Setting	Value
Basic Settings	Location	B1.P3 [a]
Sources	Sources	(Selected)
	Sources > Equation Sources	acid.mf
	Sources > Equation Sources > acid.mf	(Selected)
	Sources > Equation Sources > acid.mf > Option	Source

Tab	Setting	Value
	Sources > Equation Sources > acid.mf > Source	AcidSource
	Sources > Equation Sources > acid.mf > Source Coefficient	(Selected)
	Sources > Equation Sources > acid.mf > Source Coefficient > Source Coefficient	AcidSourceCoeff
	Sources > Equation Sources	alkali.mf
	Sources > Equation Sources > alkali.mf	(Selected)
	Sources > Equation Sources > alkali.mf > Option	Source
	Sources > Equation Sources > alkali.mf > Source	AlkaliSource
	Sources > Equation Sources > alkali.mf > Source Coefficient	(Selected)
	Sources > Equation Sources > alkali.mf > Source Coefficient > Source Coefficient	AlkaliSource-Coeff
	Sources > Equation Sources	Energy
	Sources > Equation Sources > Energy	(Selected)
	Sources > Equation Sources > Energy > Option	Source
	Sources > Equation Sources > Energy > Source	HeatSource
	Sources > Equation Sources	product.mf
	Sources > Equation Sources > product.mf	(Selected)
	Sources > Equation Sources > product.mf > Option	Source
	Sources > Equation Sources > product.mf > Source	ProductSource
	Sources > Equation Sources > product.mf > Source Coefficient	(Selected)
	Sources > Equation Sources > product.mf > Source Coefficient > Source Coefficient	0 [kg m^-3 s^-1]

[a]This is the 3D region that fills the domain.

4. Click **OK**.

15.6.7. Creating the Boundary Conditions

Add boundary conditions for all boundaries except the mixing tube wall; the latter will receive the default wall condition. Many of the required settings were given in the problem description. Since the fluid in the domain is a multicomponent fluid, you can control which component enters at each inlet by setting mass fractions appropriately. Note that water is the constraint material; its mass fraction is computed as unity minus the sum of the mass fractions of the other components.

15.6.7.1. Water Inlet Boundary

Create a boundary for the water inlet using the given information:

1. Create a new boundary named `InWater`.
2. Apply the following settings:

Tab	Setting	Value
Basic Settings	Boundary Type	Inlet
	Location	InWater
Boundary Details	Mass and Momentum > Option	Normal Speed
	Mass and Momentum > Normal Speed	2 [m s^-1]
	Heat Transfer > Option	Static Temperature
	Heat Transfer > Static Temperature	300 [K]

3. Leave mass fractions for all components set to zero. Since `Water` is the constraint fluid, it will be automatically given a mass fraction of 1 on this inlet.
4. Click **OK**.

15.6.7.2. Acid Inlet Boundary

Create a boundary for the acid solution inlet hole using the given information:

1. Create a new boundary named `InAcid`.
2. Apply the following settings:

Tab	Setting	Value
Basic Settings	Boundary Type	Inlet
	Location	InAcid
Boundary Details	Mass and Momentum > Option	Normal Speed
	Mass and Momentum > Normal Speed	2 [m s^-1]
	Heat Transfer > Option	Static Temperature
	Heat Transfer > Static Temperature	300 [K]
	Component Details	acid
	Component Details > acid > Mass Fraction	1.0
	Component Details	alkali
	Component Details > alkali > Mass Fraction	0
	Component Details	product
	Component Details > product > Mass Fraction	0

3. Click **OK**.

15.6.7.3. Alkali Inlet Boundary

Create a boundary for the alkali solution inlet holes using the given information:

1. Create a new boundary named `InAlkali`.
2. Apply the following settings:

Tab	Setting	Value
Basic Settings	Boundary Type	Inlet
	Location	InAlkali
Boundary Details	Mass and Momentum > Option	Normal Speed
	Mass and Momentum > Normal Speed	2.923 [m s^-1]
	Heat Transfer > Option	Static Temperature
	Heat Transfer > Static Temperature	300 [K]
	Component Details > acid	(Selected)
	Component Details > acid > Mass Fraction	0
	Component Details > alkali	(Selected)
	Component Details > alkali > Mass Fraction	1
	Component Details > product	(Selected)
	Component Details > product > Mass Fraction	0

3. Click **OK**.

15.6.7.4. Outlet Boundary

Create a subsonic outlet at 1 atm (which is the reference pressure that was set in the domain definition):

1. Create a new boundary named `out`.
2. Apply the following settings:

Tab	Setting	Value
Basic Settings	Boundary Type	Outlet
	Location	out
Boundary Details	Mass and Momentum > Option	Static Pressure
	Mass and Momentum > Relative Pressure	0 [Pa]

3. Click **OK**.

15.6.7.5. Symmetry Boundary

The geometry models a 30° slice of the full geometry. Create two symmetry boundaries, one for each side of the geometry, so that the simulation models the entire geometry.

1. Create a new boundary named `sym1`.
2. Apply the following settings:

Tab	Setting	Value
Basic Settings	Boundary Type	Symmetry
	Location	sym1

3. Click **OK**.
4. Create a new boundary named `sym2`.
5. Apply the following settings:

Tab	Setting	Value
Basic Settings	Boundary Type	Symmetry
	Location	sym2

6. Click **OK**.

Note that, in this case, a periodic interface can be used as an alternative to the symmetry boundary conditions.

15.6.7.6. Default Wall Boundary

The default adiabatic wall boundary applies automatically to the remaining unspecified boundary, which is the mixer wall. The default boundary is a smooth, no-slip, adiabatic wall.

15.6.8. Setting Initial Values

The values for `acid`, `alkali`, and `product` will be initialized to 0. Since `Water` is the constrained component, it will make up the remaining mass fraction which, in this case, is 1.

Since the inlet velocity is 2 m/s, a reasonable guess for the initial velocity is 2 m/s.

1. Click *Global Initialization*  .
2. Apply the following settings:

Tab	Setting	Value
Global Settings	Initial Conditions > Cartesian Velocity Components > Option	Automatic with Value
	Initial Conditions > Cartesian Velocity Components > U	2 [m s^-1]
	Initial Conditions > Cartesian Velocity Components > V	0 [m s^-1]
	Initial Conditions > Cartesian Velocity Components > W	0 [m s^-1]
	Initial Conditions > Component Details	acid
	Initial Conditions > Component Details > acid > Option	Automatic with Value
	Initial Conditions > Component Details > acid > Mass Fraction	0
	Initial Conditions > Component Details	alkali
	Initial Conditions > Component Details > alkali > Option	Automatic with Value
	Initial Conditions > Component Details > alkali > Mass Fraction	0

Tab	Setting	Value
	Initial Conditions > Component Details	product
	Initial Conditions > Component Details > product > Option	Automatic with Value
	Initial Conditions > Component Details > product > Mass Fraction	0

3. Click **OK**.

15.6.9. Setting Solver Control

1. Click *Solver Control*  .
2. Apply the following settings:

Tab	Setting	Value
Basic Settings	Advection Scheme > Option	High Resolution
	Convergence Control > Max. Iterations	50
	Convergence Control > Fluid Timescale Control > Timescale Control	Physical Timescale
	Convergence Control > Fluid Timescale Control > Physical Timescale	0.01 [s][a]

[a]The length of mixing tube is 0.06 m and inlet velocity is 2 m/s. An estimate of the dynamic time scale is 0.03 s. An appropriate time step would be 1/4 to 1/2 of this value.

3. Click **OK**.

15.6.10. Writing the CFX-Solver Input (.def) File

1. Click *Define Run*  .
2. Apply the following settings:

Setting	Value
File name	Reactor.def

3. Click **Save**.

   CFX-Solver Manager automatically starts and, on the **Define Run** dialog box, the **Solver Input File** is set.
4. If using Standalone Mode, quit CFX-Pre, saving the simulation (`.cfx`) file at your discretion.

15.7. Obtaining a Solution Using CFX-Solver Manager

When CFX-Solver Manager has started, obtain a solution to the CFD problem as follows:

1. Ensure **Define Run** is displayed.

2. Select **Show Advanced Controls**. On the **Solver** tab, select **Executable Settings** > **Double Precision Override** > **Double Precision**.

   This provides the precision required to evaluate the expression for pH.

3. Click **Start Run**.

   CFX-Solver runs and attempts to obtain a solution. This may take a long time, depending on your system. Eventually a dialog box is displayed.

4. Select the check box next to **Post-Process Results** when the completion message appears at the end of the run.

5. If using Standalone Mode, select the check box next to **Shut down CFX-Solver Manager**.

6. Click **OK**.

15.8. Viewing the Results in CFD-Post

To see the nature and extent of the reaction process, examine the pH, the mass fractions, and turbulence quantities on a plane as follows:

1. Create an XY slice plane through Z = 0.

2. Turn off the visibility of the plane you just created.

3. Create contour plots of the following variables on that plane:

   - `MixturePH`
   - `acid.Mass Fraction`
   - `alkali.Mass Fraction`
   - `product.Mass Fraction`
   - `Turbulence Kinetic Energy`
   - `Turbulence Eddy Dissipation`

4. Create an expression for `Turbulence Eddy Dissipation/Turbulence Kinetic Energy`, then create a variable using the expression (only variables can be plotted) and create a contour plot using that variable. This quantity is an indicator of the reaction rate — it represents 1 / mixing timescale.

Chapter 16: Heat Transfer from a Heating Coil

This tutorial includes:

16.1. Tutorial Features

In this tutorial you will learn about:

- Creating and using a solid domain as a heating coil in CFX-Pre.
- Creating a domain interface.
- Modeling conjugate heat transfer in CFX-Pre.
- Using electricity to power a heat source.
- Creating and using a thin-walled fluid domain in CFX-Pre.
- Modeling varying physics between multiple fluid domains.
- Plotting temperature on a cylindrical locator in CFD-Post.
- Lighting in CFD-Post.
- Exporting thermal and mechanical data to be used with ANSYS Multi-field solver.

Component	Feature	Details
CFX-Pre	User Mode	General Mode
	Analysis Type	Steady State
	Fluid Type	General Fluid
	Domain Type	Multiple Domain
	Turbulence Model	k-Epsilon
		Shear Stress Transport
	Heat Transfer	Thermal Energy
	Heat Transfer Modeling	Conjugate Heat Transfer (via Electrical Resistance Heating)

Component	Feature	Details
		Conduction Through a Thin Wall
	Boundary Conditions	Inlet (Subsonic)
		Outlet (Subsonic)
		Opening
		Wall: No-Slip
		Wall: Adiabatic
	CEL (CFX Expression Language)	
	Timestep	Physical Time Scale
CFD-Post	Plots	Contour
		Cylindrical Locator
		Isosurface
		Temperature Profile Chart
	Other	Changing the Color Range
		Expression Details View
		Lighting Adjustment
		Variable Details View
		Exporting Results to ANSYS

16.2. Overview of the Problem to Solve

The first portion of this tutorial demonstrates the capability of ANSYS CFX to model conjugate heat transfer. A simple heat exchanger is used to model the transfer of thermal energy from an electrically-heated solid copper coil to the water flowing around it. The latter section demonstrates the capability of ANSYS CFX to model heat transfer through a thin surface. The initial simulation will be altered so that the heating coil becomes a thin-walled copper tube with dry steam flowing through it.

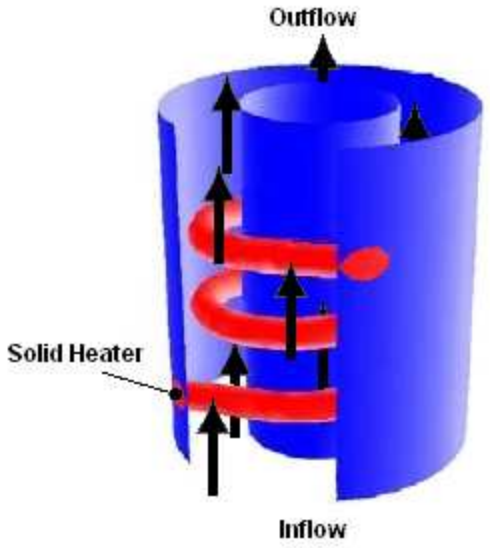
The first model contains a fluid domain for the water and a solid domain for the coil. The fluid domain is an annular region that envelops the coil, and has water at an initial temperature of 300 K flowing through it at 0.4 m/s. The copper coil has a 4.4 V difference in electric potential from one end to the other end. and is given an initial temperature of 550 K. Assume that the copper has a uniform electrical conductivity of 59.6E+06 S/m and that there is a 1 mm thick calcium carbonate deposit (calcite) on the heating coil.

The other material parameters for the calcium carbonate deposit are:

- Molar Mass = `100.087[kg kmol^-1]`
- Density = `2.71[g cm^-3]`
- Specific Heat Capacity = `0.9[J g^-1 K^-1]`
- Thermal Conductivity = `3.85[W m^-1 K^-1]`

The second model will maintain the original annular fluid domain, and turn the solid domain into a second fluid domain. Settings will be adjusted so that these two fluid domains can have separate physics. The domain interface will be modified to model a thin copper pipe, with a 1 mm thick calcium carbonate deposit, containing dry steam at an initial temperature of 600 K and an initial velocity of 0.25 m/s. The steam outlet will

have a relative pressure of 0 psi. All material properties for the dry steam will be set using the **IAPWS Library** option and using all default table values. The calcium carbonate deposit will have the same material parameters as provided above.



This tutorial also includes an optional step that demonstrates the use of the CFX to ANSYS Data Transfer tool to export thermal and mechanical stress data for use with ANSYS Multi-field solver. A results file is provided in case you want to skip the model creation and solution steps within ANSYS CFX.

16.3. Before You Begin

If this is the first tutorial you are working with, it is important to review the following topics before beginning:

- *Preparing a Working Directory and Starting ANSYS CFX in Standalone Mode* (p. 1)
- *Running ANSYS CFX Tutorials in ANSYS Workbench* (p. 2)
- *Changing the Display Colors* (p. 5)
- *Playing a Tutorial Session File* (p. 4)

16.4. Starting CFX-Pre

1. Prepare the working directory using the following files in the `examples` directory:
   - `HeatingCoil.pre`
   - `HeatingCoil.cfx`
   - `SteamCoil.pre`
   - `HeatingCoilMesh.gtm`
2. Set the working directory and start CFX-Pre

   For details, see *Preparing a Working Directory and Starting ANSYS CFX in Standalone Mode* (p. 1)

16.5. Defining a Case in CFX-Pre

If you want to set up the simulation automatically using a tutorial session file, run `HeatingCoil.pre`. For details, see *Playing a Tutorial Session File* (p. 4). Then proceed to *Obtaining a Solution using CFX-Solver Manager* (p. 271).

If you want to set up the simulation manually, proceed to the following steps:

1. In CFX-Pre, select **File** > **New Case**.
2. Select **General** and click **OK**.
3. Select **File** > **Save Case As**.
4. Under **File name**, type `HeatingCoil`.
5. If you are notified the file already exists, click **Overwrite**. This file is provided in the tutorial directory and may exist in your working directory if you have copied it there.
6. Click **Save**.

16.5.1. Importing the Mesh

1. Expand the `Case Options` section in the **Outline** tree view.
2. Edit `General`.
3. Turn off **Automatic Default Domain** and **Automatic Default Interfaces**.

   Default domain and interface generation should be turned off because you will manually create the fluid and solid domains and interface later in this tutorial.
4. Click **OK** to apply this change.
5. Right-click `Mesh` and select **Import Mesh** > **CFX Mesh**.

   The **Import Mesh** dialog box appears.
6. Apply the following settings:

Setting	Value
File name	HeatingCoilMesh.gtm

7. Click **Open**.
8. Right-click a blank area in the viewer and select **Predefined Camera** > **Isometric View (Z up)** from the shortcut menu.

16.5.2. Editing the Material Properties

1. Expand `Materials` from the tree view, right-click `Copper` and select **Edit**.
2. Apply the following settings to `Copper`:

Tab	Setting	Value
Material Properties	Electromagnetic Properties	Expand the **Electromagnetic Properties** frame [*1* **(p. 265)**]

Tab	Setting	Value
	Electromagnetic Properties > Electrical Conductivity	(Selected)
	Electromagnetic Properties > Electrical Conductivity > Electrical Conductivity	59.6E+06 [S m^-1]

Footnote

1. Expand a section by clicking *Roll Down*  .

3. Click **OK** to apply these settings to `Copper`.

16.5.3. Defining the Calcium Carbonate Deposit Material

Create a new material definition that will be used to model the calcium carbonate deposit on the heating coil:

1. Click *Material*  and name the new material `Calcium Carbonate`.
2. Apply the following settings:

Tab	Setting	Value
Basic Settings	Material Group	User [1 (p. 266)]
	Thermodynamic State	(Selected)
	Thermodynamic State > Thermodynamic State	Solid
Material Properties	Thermodynamic Properties > Equation of State > Molar Mass	100.087 [kg kmol^-1]
	Thermodynamic Properties > Equation of State > Density	2.71 [g cm^-3] [2 (p. 266)]
	Thermodynamic Properties > Specific Heat Capacity	(Selected)
	Thermodynamic Properties > Specific Heat Capacity > Specific Heat Capacity	0.9 [J g^-1 K^-1] [2 (p. 266)]
	Transport Properties > Thermal Conductivity	(Selected) [3 (p. 266)]
	Transport Properties > Thermal Conductivity > Thermal Conductivity	3.85 [W m^-1 K^-1]

Footnotes

1. The material properties for Calcium Carbonate defined in this table came directly from the *Overview of the Problem to Solve* (p. 262) section at the beginning of this tutorial.
2. Make sure that you change the units to those indicated.
3. You may need to first expand the **Transport Properties** frame by clicking *Roll Down*  .

3. Click **OK** to apply these settings.

16.5.4. Creating the Domains

This simulation requires both a fluid domain and a solid domain. First, you will create a fluid domain for the annular region of the heat exchanger.

16.5.4.1. Creating a Fluid Domain

The fluid domain will include the region of fluid flow but exclude the solid copper heater coil.

1. Ensure that `Flow Analysis 1` > `Default Domain` does not appear in the **Outline** tree view. If it does, right-click `Default Domain` and select **Delete**.
2. Click *Domain*  and set the name to `WaterZone`.
3. Apply the following settings to `WaterZone`:

Tab	Setting	Value
Basic Settings	Location and Type > Location	Annulus [*1* (p. 266)]
	Fluid and Particle Definitions	Fluid 1
	Fluid and Particle Definitions > Fluid 1 > Material	Water
	Domain Models > Pressure > Reference Pressure	1 [atm]
Fluid Models	Heat Transfer > Option	Thermal Energy
Initialization	Domain Initialization	(Selected)

Footnote

1. This region name may be different depending on how the mesh was created. You should pick the region that forms the exterior surface of the volume surrounding the coil.

4. Click **OK** to apply these settings to `WaterZone`.

16.5.4.2. Creating a Solid Domain

Since you know that the copper heating element will be much hotter than the fluid, you can initialize the temperature to a reasonable value. The initialization option that is set when creating a domain applies only to that domain.

Create the solid domain as follows:

1. Create a new domain named `SolidZone`.
2. Apply the following settings:

Tab	Setting	Value
Basic Settings	Location and Type > Location	Coil [1 (p. 267)]
	Location and Type > Domain Type	Solid Domain
	Solid Definitions	Solid 1
	Solid Definitions> Solid 1 > Solid 1 > Material	Copper
Solid Models	Heat Transfer > Option	Thermal Energy
	Electromagnetic Model	(Selected)
	Electromagnetic Model > Electric Field Model > Option	Electric Potential
Initialization	Domain Initialization > Initial Conditions > Temperature > Option	Automatic with Value
	Domain Initialization > Initial Conditions > Temperature > Temperature	550 [K]

Footnote

1. This region name may be different depending on how the mesh was created. You should pick the region that forms the coil.

3. Click **OK** to apply these settings.

16.5.5. Creating the Boundaries

You will now set the boundary conditions using the values given in the problem description.

16.5.5.1. Heating Coil Boundaries

In order to pass electricity through the heating coil, you are going to specify a voltage of 0 [V] at one end of the coil and 4.4 [V] at the other end:

1. Click *Boundary*  and select `in SolidZone` from the drop-down menu that appears.
2. Name this new boundary `Ground` and click **OK**.
3. Apply the following settings:

Tab	Setting	Value
Basic Settings	Boundary Type	Wall
	Location	Coil End 1[1 (p. 268)]
Boundary Details	Electric Field > Option	Voltage
	Electric Field > Voltage	0 [V]

Footnote

1. You will need to click *Multi-select from extended list*  to see a list of all regions.

4. Click **OK** to apply these settings.
5. Create a similar boundary named `Hot` at the other end of the coil, Coil End 2, and apply a voltage of 4.4[V].

16.5.5.2. Inlet Boundary

You will now create an inlet boundary for the cooling fluid (Water).

1. Create a new boundary in the `WaterZone` domain named `inflow`.
2. Apply the following settings:

Tab	Setting	Value
Basic Settings	Boundary Type	Inlet
	Location	inflow
Boundary Details	Mass and Momentum > Option	Normal Speed
	Mass and Momentum > Normal Speed	0.4 [m s^-1]
	Heat Transfer > Option	Static Temperature
	Heat Transfer > Static Temperature	300 [K]

3. Click **OK** to apply these settings.

16.5.5.3. Opening Boundary

An opening boundary is appropriate for the exit in this case because, at some stage during the solution, the coiled heating element will cause some recirculation at the exit. At an opening boundary you need to set the temperature of fluid that enters through the boundary. In this case it is useful to base this temperature on the fluid temperature at the outlet, since you expect the fluid to be flowing mostly out through this opening.

1. Insert a new expression by clicking *Expression*  .
2. Name this new expression `OutletTemperature` and press the **Enter** key to continue.
3. In the **Definition** entry box, type the formula `areaAve(T)@outflow`
4. Click **Apply**.

5. Close the **Expressions** view by clicking *Close*  at the top of the tree view.
6. Create a new boundary in the `WaterZone` domain named `outflow`.
7. Apply the following settings:

Tab	Setting	Value
Basic Settings	Boundary Type	Opening
	Location	outflow
Boundary Details	Mass and Momentum > Option	Opening Pres. and Dirn
	Mass and Momentum > Relative Pressure	0 [Pa]
	Heat Transfer > Option	Static Temperature
	Heat Transfer > Static Temperature	OutletTemperature [1 (p. 269)]

Footnote

1. In order to enter an expression, you need to click *Enter Expression*  .

8. Click **OK** to apply these settings.

A default no slip, adiabatic wall boundary named `WaterZone Default` will be applied automatically to the remaining unspecified external boundaries of the `WaterZone` domain.

Two more boundary conditions are generated automatically when a domain interface is created to connect the fluid and solid domains. The domain interface is discussed in the next section.

16.5.6. Creating the Domain Interface

If you have **Automatic Default Interfaces** turned on, then an interface called `Default Fluid Solid Interface` is created automatically and listed in the tree view. In this case, delete the default interface and proceed with creating a new one.

1. Click *Domain Interface*  from the row of icons located along the top of the screen.
2. Set the name to `Domain Interface` and click **OK** to accept it.
3. Apply the following settings to `Domain Interface`:

Tab	Setting	Value
Basic Settings	Interface Type	Fluid Solid
	Interface Side 1 > Domain (Filter)	WaterZone
	Interface Side 1 > Region List	coil surface
	Interface Side 2 > Domain (Filter)	SolidZone
	Interface Side 2 > Region List	F22.33, F30.33, F31.33, F32.33, F34.33, F35.33

Tab	Setting	Value
Additional Interface Models	Heat Transfer	(Selected)
	Heat Transfer > Interface Model > Option	Thin Material
	Heat Transfer > Interface Model > Material	Calcium Carbonate
	Heat Transfer > Interface Model > Thickness	1 [mm] [1 (p. 270)]

Footnote

1. Make sure that you change the units to those indicated.

4. Click **OK** to apply these settings.

16.5.7. Setting Solver Control

1. Click *Solver Control*  .
2. Apply the following settings:

Tab	Setting	Value
Basic Settings	Convergence Control > Fluid Timescale Control > Timescale Control	Physical Timescale
	Convergence Control >Fluid Timescale Control > Physical Timescale	2 [s]

For the **Convergence Criteria**, an RMS value of at least 1e-05 is usually required for adequate convergence, but the default value is sufficient for demonstration purposes.

3. Click **OK** to apply these settings.

16.5.8. Writing the CFX-Solver Input (.def) File

1. Click *Define Run*  .
2. Apply the following settings:

Setting	Value
File name	HeatingCoil.def

3. Click **Save**.

CFX-Solver Manager automatically starts and, on the **Define Run** dialog box, the **Solver Input File** is set.

4. If using Standalone Mode, quit CFX-Pre, saving the simulation (`.cfx`) file at your discretion.

16.6. Obtaining a Solution using CFX-Solver Manager

1. Ensure that the **Define Run** dialog box is displayed.
2. Click **Start Run**.

   CFX-Solver runs and attempts to obtain a solution. This can take a long time depending on your system. Eventually a dialog box is displayed.

   While the calculations proceed, you can see residual output for various equations in both the text area and the plot area. Use the tabs to switch between different plots (e.g., **Heat Transfer**, **Turbulence (KE)**, etc.) in the plot area. You can view residual plots for the fluid and solid domains separately by editing the workspace properties (under **Workspace** > **Workspace Properties**).
3. Select the check box next to **Post-Process Results** when the completion message appears at the end of the run.
4. If using Standalone Mode, select the check box next to **Shut down CFX-Solver Manager**.
5. Click **OK**.

16.7. Viewing the Results in CFD-Post

The following topics will be discussed:

16.7.1. Heating Coil Temperature Range

To grasp the effect of the calcium carbonate deposit, it is beneficial to compare the temperature range on either side of the deposit.

1. When CFD-Post opens, if you see the Domain Selector dialog box, ensure that both domains are selected, then click **OK**.
2. Create a new contour named `Contour 1`.
3. Apply the following settings:

Tab	Setting	Value
Geometry	Location	Domain Interface Side 1 [1 (p. 271)]
	Variable	Temperature
	Range	Local
	Boundary Data > Hybrid	(Selected)

Footnote

1. This is the deposit side that is in contact with the water.

4. Click **Apply**.
5. Take note of the temperature range displayed below the **Range** drop-down box. The temperature on the outer surface of the deposit should range from around 380 [K] to 740 [K].

   Change the contour location to `Domain Interface Side 2` (The deposit side that is in contact with the coil) and click **Apply**. Notice how the temperature ranges from around 420 [K] to 815 [K] on the inner surface of the deposit.

16.7.2. Creating a Cylindrical Locator

Next, you will create a cylindrical locator close to the outside wall of the annular domain. This can be done by using an expression to specify radius and locating a particular radius with an isosurface.

16.7.2.1. Expression

1. Create a new expression by clicking *Expression*  .
2. Set the name of this new expression to `expradius` and press the **Enter** key to continue.
3. Apply the following settings:

Setting	Value
Definition	(x^2 + y^2)^0.5

4. Click **Apply**.

16.7.2.2. Variable

1. Create a new variable by clicking *Variable*  .
2. Set the name of this new variable to `radius` and press the **Enter** key to continue.
3. Apply the following settings:

Setting	Value
Expression	expradius

4. Click **Apply**.

16.7.2.3. Isosurface of the variable

1. Insert a new isosurface by clicking *Location* 

>**Isosurface**.
2. Accept the default name `Isosurface 1` by clicking **OK**.
3. Apply the following settings:

Tab	Setting	Value
Geometry	Definition > Variable	radius
	Definition > Value	0.8 [m] [1 (p. 273)]
Color	Mode	Variable

Tab	Setting	Value
	Variable	Temperature
	Range	User Specified [2 (p. 273)]
	Min	299 [K]
	Max	309 [K]
Render	Show Faces	(Selected)

Footnotes

1. The maximum radius is 1 m, so a cylinder locator at a radius of 0.8 m is suitable.
2. The full temperature range is much larger due to temperature extremes on a small fraction of the isosurface. By neglecting those extreme temperatures, more colors are used over the range of interest.

4. Click **Apply**.
5. Turn off the visibility of `Contour 1` so that you have an unobstructed view of `Isosurface 1`.

Note

The default range legend now displayed is that of the isosurface and not the contour. The default legend is set according to what is being edited in the details view.

16.7.2.4. Creating a Temperature Profile Chart

For a quantitative analysis of the temperature variation through the water and heating coil, it is beneficial to create a temperature profile chart.

First, you will create a line that passes through two turns of the heating coil. You can then graphically analyze the temperature variance along that line by creating a temperature chart.

1. Insert a line by clicking *Location* 

> **Line**.
2. Accept the default name `Line 1` by clicking **OK**.
3. Apply the following settings to `Line 1`

Tab	Setting	Value
Geometry	Definition > Point 1	-0.75, 0, 0
	Definition > Point 2	-0.75, 0, 2.25
	Line Type > Sample	(Selected)
	Line Type > Samples	200

4. Click **Apply**.
5. Create a new chart by clicking *Chart*  .
6. Name this chart `Temperature Profile` and press the **Enter** key to continue.

7. Click the **Data Series** tab.
8. Set **Data Source** > **Location** to `Line 1`.
9. Click the **Y Axis** tab.
10. Set **Data Selection** > **Variable** to `Temperature`.
11. Click **Apply**.

You can see from the chart that the temperature spikes upward when entering the deposit region and is at its maximum at the center of the coil turns.

16.7.3. Specular Lighting

Specular lighting is on by default. Specular lighting allows glaring bright spots on the surface of an object, depending on the orientation of the surface and the position of the light. You can disable specular lighting as follows:

1. Click the **3D Viewer** tab at the bottom of the viewing pane.
2. Edit `Isosurface 1` in the **Outline** tree view.

Tab	Setting	Value
Render	Show Faces > Specular	(Cleared)

3. Click **Apply**.

16.7.4. Moving the Light Source

To move the light source, click within the 3-D Viewer, then press and hold **Shift** while pressing the arrow keys left, right, up or down.

Tip

If using the Standalone version, you can move the light source by positioning the mouse pointer in the viewer, holding down the **Ctrl** key, and dragging using the right mouse button.

16.8. Exporting the Results to ANSYS

This optional step involves generating an ANSYS `.cdb` data file from the results generated in CFX-Solver. The `.cdb` file could then be used with the ANSYS Multi-field solver to measure the combined effects of thermal and mechanical stresses on the solid heating coil.

There are two possible ways to export data to ANSYS:

- Use CFX-Solver Manager to export data.
- Use CFD-Post to export data. This involves:
  1. Importing a surface mesh from ANSYS into CFD-Post, and associating the surface with the corresponding 2D region in the CFX-Solver results file.
  2. Exporting the data to a file containing SFE commands that represent surface element thermal or mechanical stress values.

3. Loading the commands created in the previous step into ANSYS and visualizing the loads.

In this case, you will be using CFX-Solver Manager to export data. Since the heat transfer in the solid domain was calculated in ANSYS CFX, the 3D thermal data will be exported using element type `3D Thermal (70)`. The mechanical stresses are calculated on the liquid side of the liquid-solid interface. These values will be exported using element type `2D Stress (154)`.

16.8.1. Thermal Data

1. Start CFX-Solver Manager.
2. Select **Tools** > **Export to ANSYS MultiField**.

   The **Export to ANSYS MultiField Solver** dialog box appears.
3. Apply the following settings:

Setting	Value
Results File	HeatingCoil_001.res
Export File	HeatingCoil_001_ansysfsi_70.cdb
Domain Name > Domain	SolidZone
Domain Name > Boundary	(Empty) [1 (p. 275)]
Export Options > ANSYS Element Type	3D Thermal (70)

Footnote

1. Leave Boundary empty since the entire volume is exported for 3D data.

4. Click **Export**.

   When the export is complete, click **OK** to acknowledge the message and continue with the next steps to export data for *Mechanical Stresses* (p. 275).

16.8.2. Mechanical Stresses

1. Apply the following settings in the **Export to ANSYS MultiField Solver** dialog box:

Setting	Value
Results File	HeatingCoil_001.res
Export File	HeatingCoil_001_ansysfsi_154.cdb
Domain Name > Domain	WaterZone
Domain Name > Boundary	WaterZone Default
Export Options > ANSYS Element Type	2D Stress (154)

2. Click **Export**.

   When the export is complete, click **OK** to acknowledge the message and continue.

3. Click **Close**.
4. Close CFX-Solver Manager.

You now have two exported files that can be used with ANSYS Multi-field solver. When you are finished, close CFX-Solver Manager and CFD-Post.

16.9. Defining a Simulation Using a Thin-Walled Copper Coil

In this second part of the tutorial, you will modify the simulation from the first part of the tutorial to use a second fluid domain representing a thin-walled copper coil with dry steam running through, rather than the solid copper electric heating coil. Running the simulation a second time will demonstrate how to model multiple fluid domains with varying physics, as well as how to model heat transfer through a thin surface.

If you want to set up the steam coil simulation automatically using a tutorial session file, run `Steam-Coil.pre`. For details, see *Playing a Tutorial Session File* (p. 4). Then proceed to *Obtaining a Solution using CFX-Solver Manager* (p. 282).

1. Start CFX-Pre if it is not already running.
2. Select **File** > **Open Case**.
3. From your working directory, select `HeatingCoil.cfx` and click **Open**.
4. Select **File** > **Save Case As**.
5. Set **File name** to `SteamCoil.cfx`.
6. Click **Save**.

16.9.1. Allowing for Fluid Domains with Separate Physics and Enabling Beta Features

In this section, you will disable **Constant Domain Physics** for this case. This will enable you to create two fluid domains with separate physical settings. Since this capability is a Beta feature, you must first enable the use of Beta features.

1. Edit `Case Options` > `General` in the **Outline** tree view.
2. Select the **Physics** > **Enable Beta Features** check box.
3. Clear the **Physics** > **Constant Domain Physics** check box.
4. Click **OK** to apply this change.

Note

Make sure to make the changes via `Case Options` > `General` in the **Outline** tree instead of via the **Edit** menu (**Edit** > **Options** > **CFX-Pre** > **General**).

16.9.2. Editing Copper Properties

In this section, you will remove the electromagnetic properties of the copper (as defined in the first segment of this tutorial).

1. Edit `Materials` > `Copper` in the **Outline** tree view.
2. Apply the following settings:

Tab	Setting	Value
Material Properties	Electromagnetic Properties	Expand the **Electromagnetic Properties** frame [1 (p. 277)]
	Electromagnetic Properties > Electrical Conductivity	(Cleared)

Footnote

1. Expand a section by clicking *Roll Down*  .

3. Click **OK** to apply this change.

16.9.3. Creating a New Material

In this section, you will create a new material called `Dry Steam`. This material will represent the dry steam that is going to flow through the hollow copper coil.

1. Right-click `Materials` in the **Outline** tree view and select **Insert** > **Material** or click *Material*  .
2. Name this new material `Dry Steam` and click **OK**.
3. Apply the following settings:

Tab	Setting	Value
Basic Settings	Material Group	Dry Steam
Material Properties	Option	IAPWS Library
	Thermodynamic Properties > Table Generation	(Selected)
	Thermodynamic Properties > Table Generation > Minimum Temperature	(Selected)
	Thermodynamic Properties > Table Generation > Minimum Temperature > Min. Temperature	273.15 [K]
	Thermodynamic Properties > Table Generation > Maximum Temperature	(Selected)
	Thermodynamic Properties > Table Generation > Maximum Temperature > Max. Temperature	1000.0 [K]
	Thermodynamic Properties > Table Generation > Minimum Absolute Pressure	(Selected)
	Thermodynamic Properties > Table Generation > Minimum Absolute Pressure > Min. Absolute Pres.	1000.0 [Pa]

Tab	Setting	Value
	Thermodynamic Properties > Table Generation > Maximum Absolute Pressure	(Selected)
	Thermodynamic Properties > Table Generation > Maximum Absolute Pressure > Max. Absolute Pres.	1.0E6 [Pa]
	Thermodynamic Properties > Table Generation > Maximum Points	(Selected)
	Thermodynamic Properties > Table Generation > Maximum Points > Maximum Points	100
	Thermodynamic Properties > Table Generation > Temp. Extrapolation	(Selected)
	Thermodynamic Properties > Table Generation > Temp. Extrapolation > Activate	(Selected)
	Thermodynamic Properties > Table Generation > Pressure Extrapolation	(Selected)
	Thermodynamic Properties > Table Generation > Pressure Extrapolation > Activate	(Selected)

4. Click **OK** to apply these settings.

16.9.4. Editing the SolidZone Domain

In this section, you will modify the `SolidZone` domain to make it representative of a thin-walled steam coil.

1. Right-click `SolidZone` in the **Outline** tree view and select **Rename**.
2. Set the new domain name to `SteamZone` and press the **Enter** key to apply this new name.
3. Edit `SteamZone` in the **Outline** tree view.
4. Apply the following settings:

Tab	Setting	Value
Basic Settings	Location and Type > Domain Type	Fluid Domain
	Fluid and Particle Definitions > Fluid 1 > Material	Dry Steam
	Domain Models > Pressure > Reference Pressure	1 [atm]
Fluid Models	Heat Transfer > Option	Thermal Energy
Solver Control	Domain Solver Control	(Selected)
	Domain Solver Control > Timescale Control > Timescale Control	Physical Timescale

Tab	Setting	Value
	Domain Solver Control > Timescale Control > Physical Timescale	5.09 [s] [1 (p. 279)]

Footnote

1. The physical timescale is derived from the approximate copper pipe length (10.7 [m]) and the average rate at which the steam flows through the pipe (0.21 [m s^-1]).

5. Click **OK** to apply these changes to the `SteamZone` domain.

Note

Please note that you will see several physics errors appear in the window below the 3D viewer. These are normal because you have just defined a second fluid at the interface, and now need to modify the domain interface from type `Fluid Solid` to type `Fluid Fluid`.

16.9.5. Editing the WaterZone Domain

In this section, you will set a separate physical timescale for this domain since the physical timescales between the two fluid domains are quite different. The physical time scale of a fluid domain should be some fraction of a length scale divided by a velocity scale. For more details on this, see Physical Time Scale in the *CFX-Solver Modeling Guide*.

1. Edit `WaterZone` in the **Outline** tree view.
2. Apply the following setting:

Tab	Setting	Value
Solver Control	Domain Solver Control	(Selected)
	Domain Solver Control > Timescale Control > Timescale Control	Physical Timescale
	Domain Solver Control > Timescale Control > Physical Timescale	0.56 [s] [1 (p. 279)]

Footnote

1. The physical timescale is derived from the annular pipe length (2.25 [m]) and the rate at which the water flows through the pipe (0.4 [m s^-1]).

3. Click **OK** to apply these settings.

16.9.6. Editing the Domain Interface

In this section, you will modify the domain interface to represent a thin copper wall.

1. Edit `Domain Interface` in the **Outline** tree view.
2. Apply the following settings:

Tab	Setting	Value
Basic Settings	Interface Type	Fluid Fluid
	Interface Side 1 > Domain (Filter)	WaterZone
	Interface Side 1 > Region List	coil surface
	Interface Side 2 > Domain (Filter)	SteamZone
	Interface Side 2 > Region List	F22.33, F30.33, F31.33, F32.33, F34.33, F35.33 [*1* **(p. 280)**]
Additional Interface Models	Mass and Momentum > Option	No Slip Wall
	Heat Transfer > Material	Copper
	Heat Transfer > Thickness	2 [mm] [*2* **(p. 280)**]

Footnote

1. Click *Multi-select from extended list*  and hold down the **Ctrl** key while selecting each of the listed regions
2. This thickness is based on the Nominal Pipe Size for a pipe with a 100 mm diameter.

3. Click **OK** to apply these changes to the domain interface.

16.9.7. Editing the Ground Boundary

In this section, you will edit the `Ground` boundary associated with the `SteamZone` domain. This boundary is currently set up as ground for the electrical heating coil model, and needs to be modified to represent the steam coil inlet.

1. Right-click the `Ground` boundary in the **Outline** tree view and select **Rename**.
2. Set the name of this boundary to `SteamIn` and press the **Enter** key to apply this change.
3. Edit `SteamIn` in the **Outline** tree view.
4. Apply the following settings:

Tab	Setting	Value
Basic Settings	Boundary Type	Inlet
Boundary Details	Mass and Momentum > Option	Cart. Vel. Components
	Mass and Momentum > U	0.25 [m s^-1]
	Mass and Momentum > V	0 [m s^-1]
	Mass and Momentum > W	0 [m s^-1]
	Heat Transfer > Static Temperature	600 [K]

Note

The values in this table come directly from the overview of the problem at the beginning of this tutorial.

5. Click **OK** to apply these changes.

16.9.8. Editing the Hot Boundary

In this section, you will edit the `Hot` boundary associated with the `SteamZone` domain. This boundary is currently set up with electric potential for the electrical heating coil model, and needs to be modified to represent the steam coil outlet.

1. Right-click the `Hot` boundary in the **Outline** tree view and select **Rename**.
2. Set the name of this boundary to `SteamOut` and press the **Enter** key to apply the change.
3. Edit `SteamOut` in the **Outline** tree view.
4. Apply the following settings:

Tab	Setting	Value
Basic Settings	Boundary Type	Outlet
Boundary Details	Mass and Momentum > Relative Pressure	0 [Pa]

5. Click **OK** to apply these changes.

16.9.9. Writing the CFX-Solver Input (.def) File

1. Click *Define Run*  .
2. Apply the following settings:

Setting	Value
File name	SteamCoil.def

3. Click **Save**.

   This tutorial makes use of a Beta feature: domain-specific solver control. A dialog box asks if you want to write the case even though it uses a Beta feature.
4. In the **Beta Physics Model Warning** dialog box, click **Yes**.

   CFX-Solver Manager automatically starts and, on the **Define Run** dialog box, the **Solver Input File** is set.

Note

If you have used the global options instead of the case options to enable the beta features, make sure to turn it off because it will cause instability in future sessions.

5. If using Standalone Mode, quit CFX-Pre, saving the simulation (`.cfx`) file at your discretion.

16.10. Obtaining a Solution using CFX-Solver Manager

1. Ensure that the **Define Run** dialog box is displayed.
2. Click **Start Run**.

   CFX-Solver runs and attempts to obtain a solution. This can take a long time depending on your system. Eventually a dialog box is displayed.

   While the calculations proceed, you can see residual output for various equations in both the text area and the plot area. Use the tabs to switch between different plots (e.g., **Heat Transfer**, **Turbulence (KE)**, etc.) in the plot area. You can view residual plots for the fluid and solid domains separately by editing the workspace properties (under **Workspace** > **Workspace Properties**).
3. Select the check box next to **Post-Process Results** when the completion message appears at the end of the run.
4. If using Standalone Mode, select the check box next to **Shut down CFX-Solver Manager**.
5. Click **OK**.

16.11. Viewing the Results in CFD-Post

The following topics will be discussed:

- *Heating Coil Temperature Range* (p. 282)
- *Creating a Cylindrical Locator* (p. 272)

16.11.1. Heating Coil Temperature Range

To examine how the heat transfer through the pipe changes along the length of the coil, it is useful to look at temperature contour along the outer surface of the coil.

1. When CFD-Post opens, if you see the Domain Selector dialog box, ensure that both domains are selected, then click **OK**.
2. Create a new contour named `Contour 1`.
3. Apply the following settings:

Tab	Setting	Value
Geometry	Location	Domain Interface Side 1 [*1* (p. 282)]
	Variable	Temperature
	Range	Local
	Boundary Data > Hybrid	(Selected)

Footnote

1. This is the deposit side that is in contact with the water.

4. Click **Apply**.
5. Take note of the temperature range displayed below the **Range** drop-down box. The temperature on the outer surface of the deposit should range from around 370 [K] to 540 [K].

   Change the contour location to `Domain Interface Side 2` (The coil inner coil surface that is in direct contact with the steam) and click **Apply**. Notice how the temperature ranges from around 370 [K] to 600 [K] on the inner surface of the coil.

16.11.2. Creating a Cylindrical Locator

Next, you will create a cylindrical locator close to the outside wall of the annular domain. This can be done by using an expression to specify radius and locating a particular radius with an isosurface.

16.11.2.1. Expression

1. Create a new expression by clicking *Expression*  .
2. Set the name of this new expression to `expradius` and press **Enter** to continue.
3. Apply the following settings:

Setting	Value
Definition	(x^2 + y^2)^0.5

4. Click **Apply**.

16.11.2.2. Variable

1. Create a new variable by clicking *Variable*  .
2. Set the name of this new variable to `radius` and press the **Enter** key to continue.
3. Apply the following settings:

Setting	Value
Expression	expradius

4. Click **Apply**.

16.11.2.3. Isosurface of the variable

1. Insert a new isosurface by clicking *Location* 

>**Isosurface**.
2. Accept the default name `Isosurface 1` by clicking **OK**.
3. Apply the following settings:

Tab	Setting	Value
Geometry	Definition > Variable	radius
	Definition > Value	0.8 [m] [1 (p. 284)]
Color	Mode	Variable

Tab	Setting	Value
	Variable	Temperature
	Range	Local

Footnote

1. The maximum radius is 1 [m], so a cylinder locator at a radius of 0.8 [m] is suitable.

4. Click **Apply**.
5. Turn off the visibility of `Contour 1` so that you have an unobstructed view of `Isosurface 1`.

You can see how the temperature along the steam coil gradually decreases along the length of the coil (as some of the heat in the steam is lost via heat transfer through the thin copper wall and to the cooler water on the other side).

Now, you will adjust the temperature range along this isosurface to get a better understanding of the heat transfer from the steam coil to the surrounding water.

- Adjust the settings of `Isosurface 1` as follows:

Tab	Setting	Value
Color	Range	User Specified
	Min	299 [K]
	Max	309 [K]

Note

The default range legend now displayed is that of the isosurface and not the contour. The default legend is set according to what is being edited in the details view.

You can see how the cool water is heated as it passes directly past the steam coil (the cool water maintains a steady temperature until it reaches the first loop in the coil).

16.11.2.4. Creating a Temperature Profile Chart

For a quantitative analysis of the temperature variation through the water and steam coil, it is beneficial to create a temperature profile chart.

First, you will create a line that passes through two turns of the heating coil. You can then graphically analyze the temperature variance along that line by creating a temperature chart.

1. Insert a line by clicking *Location* 

> **Line**.
2. Accept the default name `Line 1` by clicking **OK**.
3. Apply the following settings to `Line 1`

Tab	Setting	Value
Geometry	Definition > Point 1	-0.75, 0, 0

Tab	Setting	Value
	Definition > Point 2	-0.75, 0, 2.25
	Line Type > Sample	(Selected)
	Line Type > Samples	200

4. Click **Apply**.
5. Create a new chart by clicking *Chart*  .
6. Name this chart `Temperature Profile` and press the **Enter** key to continue.
7. Click the **Data Series** tab.
8. Set **Data Source** > **Location** to `Line 1`.
9. Click the **Y Axis** tab.
10. Set **Data Selection** > **Variable** to `Temperature`.
11. Click **Apply**.

You can see from the chart that the temperature spikes upward when entering the coil region and remains relatively steady across the cross-section of the coil.

Chapter 17: Multiphase Flow in a Mixing Vessel

This tutorial includes:

17.1. Tutorial Features

In this tutorial you will learn about:

- Setting up a multiphase flow simulation involving air and water.
- Importing meshes that have CFX-4 and CFX Mesh file formats.
- Setting up a simulation using multiple frames of reference.
- Using a fluid dependent turbulence model to set different turbulence options for each fluid.
- Specifying buoyant flow.
- Specifying a degassing outlet boundary to allow air, but not water, to escape from the boundary.
- Connecting two domains (one for a tank and one for an impeller inside the tank) via Frozen Rotor interfaces.
- Modeling rotational periodicity using periodic boundary conditions.
- Using periodic GGI interfaces where the mesh does not match exactly.
- Using thin surfaces for blade and baffle surfaces.

Component	Feature	Details
CFX-Pre	User Mode	General Mode
	Analysis Type	Steady State
	Fluid Type	General Fluid
	Domain Type	Multiple Domain
		Rotating Frame of Reference
	Turbulence Model	Dispersed Phase Zero Equation
		Fluid-Dependent
		k-Epsilon
	Heat Transfer	None
	Buoyant Flow	

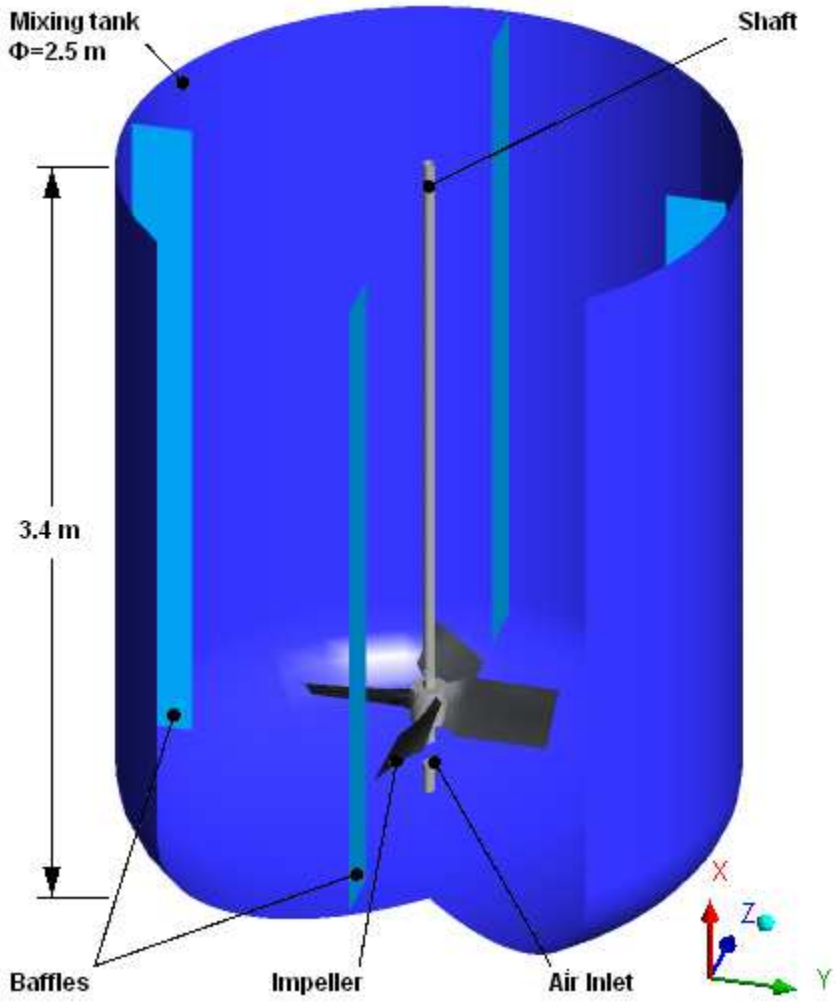
Component	Feature	Details
	Multiphase	
	Boundary Conditions	Inlet (Subsonic)
		Outlet (Degassing)
		Wall: Thin Surface
		Wall: (Slip Depends on Volume Fraction)
	Domain Interfaces	Frozen Rotor
		Periodic
		Thin Surface Partners
	Output Control	
	Timestep	Physical Time Scale
CFD-Post	Plots	Default Locators
		Slice Plane
	Other	Quantitative Calculation

17.2. Overview of the Problem to Solve

This example simulates the mixing of water and air in a mixing vessel. The geometry consists of a mixing tank vessel, an air injection pipe, four baffles, a rotating impeller, and a shaft that runs vertically through the vessel. The impeller rotates at 84 rpm about the X-axis (in the counterclockwise direction, when viewed from above). Air is injected into the vessel through an inlet pipe located below the impeller at a speed of 5 m/s. The inlet pipe diameter is 2.48 cm. Assume that both the water and air remain at a constant temperature of 25°C and that the air is incompressible, with a density equal to that at 25°C and 1 atmosphere. Also assume that the air bubbles are 3 mm in diameter.

Examine the steady-state distribution of air in the tank. Also calculate the torque and power required to turn the impeller at 84 rpm.

Figure 17.1 Cut-away Diagram of the Mixer



The figure above shows the full geometry with part of the tank walls and one baffle cut away. The symmetry of the vessel allows a 1/4 section of the full geometry to be modeled.

17.3. Before You Begin

If this is the first tutorial you are working with, it is important to review the following topics before beginning:

- *Preparing a Working Directory and Starting ANSYS CFX in Standalone Mode* (p. 1)
- *Running ANSYS CFX Tutorials in ANSYS Workbench* (p. 2)
- *Changing the Display Colors* (p. 5)
- *Playing a Tutorial Session File* (p. 4)

17.4. Starting CFX-Pre

1. Prepare the working directory using the following files in the `examples` directory:
   - `MultiphaseMixer.pre`
   - `MixerImpellerMesh.gtm`
   - `MixerTank.geo`

2. Set the working directory and start CFX-Pre.

   For details, see *Preparing a Working Directory and Starting ANSYS CFX in Standalone Mode* (p. 1)

17.5. Defining a Case in CFX-Pre

If you want to set up the simulation automatically using a tutorial session file, run `MultiphaseMixer.pre`. For details, see *Playing a Tutorial Session File* (p. 4). Then proceed to *Obtaining a Solution Using CFX-Solver Manager* (p. 304).

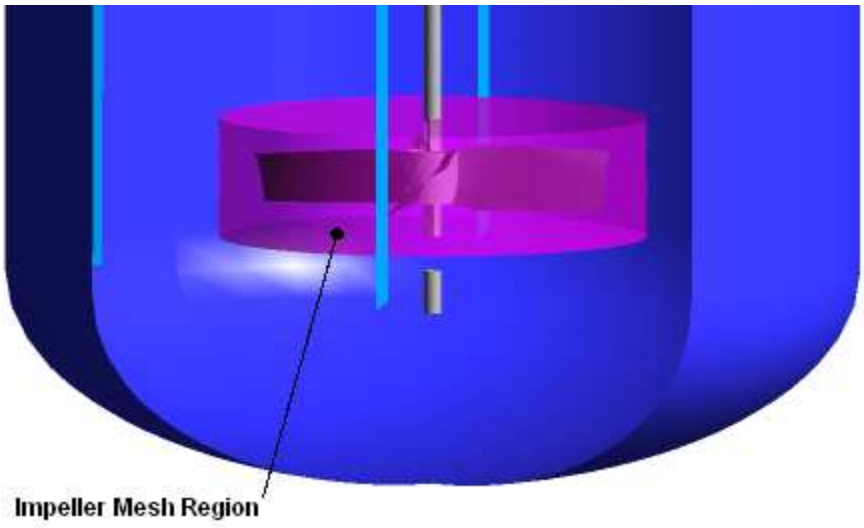
If you want to set up the simulation manually, proceed to the following steps:

1. In CFX-Pre, select **File** > **New Case**.
2. Select **General** and click **OK**.
3. Select **File** > **Save Case As**.
4. Under **File name**, type `MultiphaseMixer`.
5. Click **Save**.

17.5.1. Importing the Meshes

In this tutorial, two mesh files are provided: one for the mixer tank excluding the impeller, and one for the impeller. These meshes fit together to occupy the entire tank. The region occupied by the impeller mesh is indicated in *Figure 17.2* (p. 290).

Figure 17.2 Impeller Mesh Region



Next, you will import the mesh for the mixer tank, followed by the mesh for the impeller. The impeller mesh, as provided, is not located in the correct spatial position relative to the tank mesh. After importing the impeller mesh, you will move it to the correct position.

Note

This simulation involves the use of two domains: a stationary fluid domain on the main 3D region of the tank mesh and a rotating fluid domain on the main 3D region of the impeller mesh. It is not necessary to use separate meshes in this type of simulation, as long as there are 3D regions available for locating these two domains.

17.5.1.1. Importing the Mixer Tank Mesh

The mixer tank mesh is provided as a CFX-4 mesh file (`*.geo`). Import it as follows:

1. Edit `Case Options` > `General` in the **Outline** tree view and ensure that **Automatic Default Domain** and **Automatic Default Interfaces** are turned off.

   Default Domain and Interface generation should be turned off because you will be manually creating domains and interfaces for the impeller and main tank later in this tutorial.

2. Right-click `Mesh` and select **Import Mesh** > **Other**.

   The **Import Mesh** dialog box appears.

3. Apply the following settings:

Setting	Value
Files of type	CFX-4(*geo)
File name	MixerTank.geo
Options > Mesh Units	m
Advanced Options > CFX-4 Options > Create 3D Regions on > Fluid Regions (USER3D, POROUS)	(Cleared)[a]

[a]In this case, the mesh file contains USER3D regions that you do not need.

4. Click **Open**.

17.5.1.2. Importing the Impeller Mesh

The impeller mesh is provided as a CFX Mesh file (`*.gtm`). Import it as follows:

1. Right-click `Mesh` and select **Import Mesh** > **Other** to import the second mesh. The **Import Mesh** dialog box appears.
2. Apply the following settings:

Setting	Value
Files of type	CFX Mesh (*gtm *cfx)
File name	MixerImpellerMesh.gtm

3. Click **Open**.
4. Right-click a blank area in the viewer and select **Predefined Camera** > **Isometric View (X up)** to view the mesh assemblies.

17.5.1.3. Relocating the Impeller Mesh

In the next step you will move the impeller mesh to its correct position.

1. Right-click `MixerImpellerMesh.gtm` and select **Transform Mesh**.

   The **Mesh Transformation Editor** dialog box appears.

2. Apply the following settings:

Setting	Value
Transformation	Translation
Method	Deltas
Dx, Dy, Dz	0.275, 0, 0

3. Click **Apply** then **Close**.

17.5.1.4. Viewing the Mesh at the Tank Periodic Boundary

1. In the **Outline** workspace, expand the tree to show `MixerTank.geo` > `Principal 3D Regions` > `Primitive 3D` > `Principal 2D Regions`.
2. Click the primitive region BLKBDY_TANK_PER2.

You can now see the mesh on one of the periodic regions of the tank. To reduce the solution time for this tutorial, the mesh used is very coarse. This is not a suitable mesh to obtain accurate results, but it is sufficient for demonstration purposes.

Note

If you do not see the surface mesh, highlighting may be turned off. If highlighting is disabled, toggle *Highlighting*  . The default highlight type will show the surface mesh for any selected regions. If you see a different highlighting type, you can alter it by selecting **Edit** > **Options** and browsing to **CFX-Pre** > **Graphics Style**.

17.5.2. Creating the Domains

The mixer requires two domains: a rotating impeller domain and a stationary tank domain. Both domains contain water as a continuous phase and air as a dispersed phase. The domains will model turbulence, buoyancy, and forces between the fluids.

17.5.2.1. Rotating Domain for the Impeller

As stated in the problem description, the impeller rotates at 84 rpm.

1. Ensure that no default domain is present under `Flow Analysis 1`. If a default domain is present, right-click it and select **Delete**.
2. Click *Domain*  and set the name to `impeller`.
3. Under the **Fluid and Particle Definitions** setting, delete `Fluid 1`.
4. Click *Add new item*  and name it `Air`

5. Click *Add new item*  and name it `Water`

6. Apply the following settings:

Tab	Setting	Value
Basic Settings	Location and Type > Location	Main
	Fluid and Particle Definitions	Air
	Fluid and Particle Definitions > Air > Material	Air at 25 C
	Fluid and Particle Definitions > Air > Morphology > Option	Dispersed Fluid
	Fluid and Particle Definitions > Air > Morphology > Mean Diameter	3 [mm]
	Fluid and Particle Definitions	Water
	Fluid and Particle Definitions > Water > Material	Water
	Domain Models > Pressure > Reference Pressure	1 [atm]
	Domain Models > Buoyancy > Option	Buoyant
	Domain Models > Buoyancy > Gravity X Dirn.	-9.81 [m s^-2]
	Domain Models > Buoyancy > Gravity Y Dirn.	0 [m s^-2]
	Domain Models > Buoyancy > Gravity Z Dirn.	0 [m s^-2]
	Domain Models > Buoyancy > Buoy. Ref. Density[a]	997 [kg m^-3]
	Domain Models > Domain Motion > Option	Rotating
	Domain Models > Domain Motion > Angular Velocity	84 [rev min ^-1][b]
	Domain Models > Domain Motion > Axis Definition > Rotation Axis	Global X
Fluid Models	Multiphase > Homogeneous Model	(Cleared)[c]
	Multiphase > Free Surface Model > Option	None
	Heat Transfer > Homogeneous Model	(Cleared)
	Heat Transfer > Option	Isothermal
	Heat Transfer > Fluid Temperature	25 [C]
	Turbulence > Homogeneous Model	(Cleared)
	Turbulence > Option	Fluid Dependent
Fluid Pair Models	Fluid Pair	Air | Water
	Fluid Pair > Air | Water > Surface Tension Coefficient	(Selected)
	Fluid Pair > Air | Water > Surface Tension Coefficient > Surf. Tension Coeff.	0.073 [N m^-1][d]
	Fluid Pair > Air | Water > Momentum Transfer > Drag Force > Option	Grace
	Fluid Pair > Air | Water > Momentum Transfer > Drag Force > Volume Fraction Correction Exponent	(Selected)

Tab	Setting	Value
	Fluid Pair > Air | Water > Momentum Transfer > Drag Force > Volume Fraction Correction Exponent > Value	4[e]
	Fluid Pair > Air | Water > Momentum Transfer > Non-drag forces > Turbulent Dispersion Force > Option	Lopez de Bertodano
	Fluid Pair > Air | Water > Momentum Transfer > Non-drag forces > Turbulent Dispersion Force > Dispersion Coeff.	0.1
	Fluid Pair > Air | Water > Turbulence Transfer > Option	Sato Enhanced Eddy Viscosity[f]

[a]For dilute dispersed multiphase flow, always set the buoyancy reference density to that for continuous fluid.
[b]Note the unit.
[c]Turn off the homogeneous model to allow each fluid to have its own velocity field.
[d]This must be set to allow the Grace drag model to be used.
[e]A positive value is appropriate for large bubbles. For details, see Densely Distributed Fluid Particles: Grace Drag Model.
[f]This models particle-induced turbulence. For details, see Turbulence Enhancement.

7. Click **OK**.

17.5.2.2. Stationary Domain for the Main Tank

Next, you will create a stationary domain for the main tank by copying the properties of the existing impeller domain.

1. Right-click `impeller` and select **Duplicate** from the shortcut menu.
2. Rename the duplicated domain to `tank` and then open it for editing.
3. Apply the following settings:

Tab	Setting	Value
Basic Settings	Location and Type > Location	Primitive 3D
	Domain Models > Domain Motion > Option	Stationary

4. Click **OK**.

17.5.3. Creating the Boundaries

The following boundary conditions will be set:

- An inlet through which air enters the mixer.
- A degassing outlet, so that only the gas phase can leave the domain.
- Thin surfaces for the baffle.
- A wall for the hub and the portion of the shaft that is in the rotating domain. This wall will be rotating, and therefore stationary relative to the rotating domain.
- A wall for the portion of the shaft in the stationary domain. This wall will be rotating relative to the stationary domain.

When the default wall boundary is generated, the internal 2D regions of an imported mesh are ignored, while the regions that form domain boundaries are included.

Note

The blade surfaces of the impeller will be modeled using domain interfaces later in the tutorial.

17.5.3.1. Air Inlet Boundary

1. Create a new boundary in the domain `tank` named `Airin`.
2. Apply the following settings:

Tab	Setting	Value
Basic Settings	Boundary Type	Inlet
	Location	INLET_DIPTUBE
Boundary Details	Mass And Momentum > Option	Fluid Dependent
Fluid Values	Boundary Conditions	Air
	Boundary Conditions > Air > Velocity > Option	Normal Speed
	Boundary Conditions > Air > Velocity > Normal Speed	5 [m s^-1]
	Boundary Conditions > Air > Volume Fraction > Option	Value
	Boundary Conditions > Air > Volume Fraction > Volume Fraction	1
	Boundary Conditions	Water
	Boundary Conditions > Water > Velocity > Option	Normal Speed
	Boundary Conditions > Water > Velocity > Normal Speed	5 [m s^-1]
	Boundary Conditions > Water > Volume Fraction > Option	Value
	Boundary Conditions > Water > Volume Fraction > Volume Fraction	0

3. Click **OK**.

17.5.3.2. Degassing Outlet Boundary

Create a degassing outlet to represent the free surface where air bubbles escape. The continuous phase (water) sees this boundary as a free-slip wall and does not leave the domain. The dispersed phase (air) sees this boundary as an outlet.

1. Create a new boundary in the domain `tank` named `LiquidSurface`.
2. Apply the following settings:

Tab	Setting	Value
Basic Settings	Boundary Type	Outlet
	Location	WALL_LIQUID_SURFACE
Boundary Details	Mass And Momentum > Option	Degassing Condition

3. Click **OK**.

Note that no pressure is specified for this boundary. The solver will compute a pressure distribution on this fixed-position boundary to represent the surface height variations that would occur in the real flow.

17.5.3.3. Thin Surface for the Baffle

In CFX-Pre, thin surfaces can be created by specifying wall boundary conditions on both sides of internal 2D regions. Both sides of the baffle regions will be specified as walls in this case.

1. Create a new boundary in the domain `tank` named `Baffle`.
2. Apply the following settings:

Tab	Setting	Value
Basic Settings	Boundary Type	Wall
	Location	WALL_BAFFLES[a]
Boundary Details	Mass And Momentum > Option	Fluid Dependent
	Wall Contact Model > Option	Use Volume Fraction
Fluid Values	Boundary Conditions	Air
	Boundary Conditions > Air> Mass and Momentum > Option	Free Slip Wall[b]
	Boundary Conditions	Water
	Boundary Conditions > Water > Mass and Momentum > Option	No Slip Wall

[a]The `WALL_BAFFLES` region includes the surfaces on both sides of the baffle (you can confirm this by examining `WALL_BAFFLES` in the region selector).

[b]The `Free Slip Wall` condition can be used for the gas phase since the contact area with the walls is near zero for low gas phase volume fractions.

3. Click **OK**.

17.5.3.4. Wall Boundary for the Shaft

You will now set up a boundary for the portions of the shaft that are in the tank domain. Since the tank domain is not rotating, you need to specify a moving wall on the shaft to account for the shaft's rotation.

Part of the shaft is located directly above the air inlet, so the volume fraction of air in this location will be high and the assumption of zero contact area for the gas phase is not physically correct. In this case, a no slip boundary is more appropriate than a free slip condition for the air phase. When the volume fraction of air in contact with a wall is low, a free slip condition is more appropriate for the air phase.

In cases where it is important to correctly model the dispersed phase slip properties at walls for all volume fractions, you can declare both fluids as no slip, but set up an expression for the dispersed phase wall area fraction. The expression should result in an area fraction of zero for dispersed phase volume fractions from 0 to 0.3, for example, and then linearly increase to an area fraction of 1 as the volume fraction increases to 1.

1. Create a new boundary in the domain `tank` named `TankShaft`.
2. Apply the following settings:

Tab	Setting	Value
Basic Settings	Boundary Type	Wall
	Location	WALL_SHAFT, WALL_SHAFT_CENTER
Boundary Details	Mass and Momentum > Option	Fluid Dependent
	Wall Contact Model > Option	Use Volume Fraction
Fluid Values	Boundary Conditions	Air
	Boundary Conditions > Air > Mass And Momentum > Option	No Slip Wall
	Boundary Conditions > Air > Mass And Momentum > Wall Velocity	(Selected)
	Boundary Conditions > Air > Mass And Momentum > Wall Velocity > Option	Rotating Wall
	Boundary Conditions > Air > Mass And Momentum > Wall Velocity > Angular Velocity	84 [rev min ^-1][a]
	Boundary Conditions > Air > Mass And Momentum > Wall Velocity > Axis Definition > Option	Coordinate Axis
	Boundary Conditions > Air > Mass And Momentum > Wall Velocity > Axis Definition > Rotation Axis	Global X

[a]Note the unit.

3. Select `Water` and set the same values as for `Air`.
4. Click **OK**.

17.5.3.5. Required Boundary in the Impeller Domain

1. Create a new boundary in the domain `impeller` named `HubShaft`.
2. Apply the following settings:

Tab	Setting	Value
Basic Settings	Boundary Type	Wall
	Location	Hub, Shaft

Tab	Setting	Value
Boundary Details	Mass And Momentum > Option	Fluid Dependent
	Wall Contact Model > Option	Use Volume Fraction
Fluid Values	Boundary Conditions	Air
	Boundary Conditions > Air > Mass And Momentum > Option	Free Slip Wall
	Boundary Conditions	Water
	Boundary Conditions > Water > Mass and Momentum > Option	No Slip Wall

3. Click **OK**.

17.5.3.6. Modifying the Default Wall Boundary

As mentioned previously, when the volume fraction of air in contact with a wall is low, a free slip condition is more appropriate for the air phase.

1. In the tree view, open `tank Default` for editing.
2. Apply the following settings:

Tab	Setting	Value
Boundary Details	Mass and Momentum > Option	Fluid Dependent
	Wall Contact Model > Option	Use Volume Fraction
Fluid Values	Boundary Conditions	Air
	Boundary Conditions > Air > Mass And Momentum > Option	Free Slip Wall
	Boundary Conditions	Water
	Boundary Conditions > Water > Mass And Momentum > Option	No Slip Wall

3. Click **OK**.

   It is not necessary to set the default boundary in the impeller domain since the remaining surfaces will be assigned interface conditions in the next section.

17.5.4. Creating the Domain Interfaces

The following interfaces will be set:

- Blade thin surface interface.
- Rotational periodic domain interfaces for the periodic faces of the tank and impeller.
- Frozen Rotor interfaces between the impeller and tank domains.

17.5.4.1. Modeling the Blade Using a Domain Interface

You can model thin surfaces using either wall boundaries or domain interfaces. There are some differences between domain interfaces and ordinary wall boundaries; for example, CFX-Pre automatically detects the matching domain boundary regions when setting up a domain interface.

Previously, the thin surface representation of the tank baffle was modeled using boundary conditions. For demonstrational purposes, you will use a domain interface to model the thin surface representation of the impeller blade (even though using wall boundary conditions would also work).

1. Create a new domain interface named `Blade Thin Surface`.
2. Apply the following settings:

Tab	Setting	Value
Basic Settings	Interface Type	Fluid Fluid
	Interface Side 1 > Domain (filter)	impeller
	Interface Side 1 > Region List	Blade
	Interface Side 2 > Domain (filter)	impeller
	Interface Side 2 > Region List	Solid 3.3 2, Solid 3.6 2
Additional Interface Models	Mass And Momentum > Option	Side Dependent[a]

[a]This is done so that we can set a fluid dependent treatment on each side of the interface.

3. Click **OK**.

   Two boundaries named `Blade Thin Surface Side 1` and `Blade Thin Surface Side 2` are created automatically.
4. In the tree view, open `Blade Thin Surface Side 1` for editing.
5. Apply the following settings:

Tab	Setting	Value
Boundary Details	Mass and Momentum > Option	Fluid Dependent
	Wall Contact Model > Option	Use Volume Fraction
Fluid Values	Boundary Conditions	Air
	Boundary Conditions > Air > Mass And Momentum > Option	Free Slip Wall
	Boundary Conditions	Water
	Boundary Conditions > Water > Mass And Momentum > Option	No Slip Wall

6. Click **OK**.
7. In the tree view, open `Blade Thin Surface Side 2` for editing.
8. Apply the same settings as for `Blade Thin Surface Side 1`.

17.5.4.2. Rotational Periodic Interfaces

Periodic domain interfaces can either be one-to-one or GGI interfaces. One-to-one transformations occur for topologically similar meshes whose nodes match within a given tolerance. One-to-one periodic interfaces are more accurate and reduce CPU and memory requirements. Here, you will choose the `Automatic` mesh connection method, to let ANSYS CFX choose between one-to-one and GGI. For details, see Mesh Connection Options.

1. Create a new domain interface named `ImpellerPeriodic`.
2. Apply the following settings:

Tab	Setting	Value
Basic Settings	Interface Type	Fluid Fluid
	Interface Side 1 > Domain (Filter)	impeller
	Interface Side 1 > Region List	Periodic1
	Interface Side 2 > Domain (Filter)	impeller
	Interface Side 2 > Region List	Periodic2
	Interface Models > Option	Rotational Periodicity
	Interface Models > Axis Definition > Option	Coordinate Axis
	Interface Models > Axis Definition > Rotation Axis	Global X
	Mesh Connection Method > Mesh Connection > Option	Automatic

3. Click **OK**.

1. Create a new domain interface named `TankPeriodic`.
2. Apply the following settings:

Tab	Setting	Value
Basic Settings	Interface Type	Fluid Fluid
	Interface Side 1 > Domain (Filter)	tank
	Interface Side 1 > Region List	BLKBDY_TANK_PER1
	Interface Side 2 > Domain (Filter)	tank
	Interface Side 2 > Region List	BLKBDY_TANK_PER2
	Interface Models > Option	Rotational Periodicity
	Interface Models > Axis Definition > Option	Coordinate Axis
	Interface Models > Axis Definition > Rotation Axis	Global X
	Mesh Connection Method > Mesh Connection > Option	Automatic

3. Click **OK**.

17.5.4.3. Frozen Rotor Interfaces

You will now create three Frozen Rotor interfaces for the regions connecting the two domains. In this case three separate interfaces are created. You should not try to create a single domain interface for multiple surfaces that lie in different planes.

1. Create a new domain interface named `Top`.
2. Apply the following settings:

Tab	Setting	Value
Basic Settings	Interface Type	Fluid Fluid
	Interface Side 1 > Domain (Filter)	impeller
	Interface Side 1 > Region List	Top
	Interface Side 2 > Domain (Filter)	tank
	Interface Side 2 > Region List	BLKBDY_TANK_TOP
	Interface Models > Option	General Connection
	Interface Models > Frame Change/Mixing Model > Option	Frozen Rotor

3. Click **OK**.
4. Create a new domain interface named `Bottom`.
5. Apply the following settings:

Tab	Setting	Value
Basic Settings	Interface Type	Fluid Fluid
	Interface Side 1 > Domain (Filter)	impeller
	Interface Side 1 > Region List	Bottom
	Interface Side 2 > Domain (Filter)	tank
	Interface Side 2 > Region List	BLKBDY_TANK_BOT
	Interface Models > Option	General Connection
	Interface Models > Frame Change/Mixing Model > Option	Frozen Rotor

6. Click **OK**.
7. Create a new domain interface named `Outer`.
8. Apply the following settings:

Tab	Setting	Value
Basic Settings	Interface Type	Fluid Fluid
	Interface Side 1 > Domain (Filter)	impeller
	Interface Side 1 > Region List	Outer
	Interface Side 2 > Domain (Filter)	tank
	Interface Side 2 > Region List	BLKBDY_TANK_OUTER

Tab	Setting	Value
	Interface Models > Option	General Connection
	Interface Models > Frame Change/Mixing Model > Option	Frozen Rotor

9. Click **OK**.

For more details about the Frozen Rotor interface, see Frozen Rotor.

17.5.5. Setting Initial Values

You will set the initial volume fraction of air to 0, and allow the initial volume fraction of water to be computed automatically. Since the volume fractions must sum to unity, the initial volume fraction of water will be 1.

It is important to understand how the velocity is initialized in this tutorial. Here, both fluids use `Automatic` for the **Cartesian Velocity Components** option. When the `Automatic` option is used, the initial velocity field will be based on the velocity values set at inlets, openings, and outlets. In this tutorial, the only boundary that has a set velocity value is the inlet, which specifies a velocity of 5 [m s^-1] for both phases. Without setting the **Velocity Scale** parameter, the resulting initial guess would be a uniform velocity of 5 [m s^-1] in the X-direction throughout the domains for both phases. This is clearly not suitable since the water phase is enclosed by the tank. When the boundary velocity conditions are not representative of the expected domain velocities, the **Velocity Scale** parameter should be used to set a representative domain velocity. In this case the velocity scale for water is set to zero, causing the initial velocity for the water to be zero. The velocity scale is not set for air, resulting in an initial velocity of 5 [m s^-1] in the X-direction for the air. This should not be a problem since the initial volume fraction of the air is zero everywhere.

1. Click *Global Initialization*  .
2. Apply the following settings:

Tab	Setting	Value
Fluid Settings	Fluid Specific Initialization	Air
	Fluid Specific Initialization > Air > Initial Conditions > Volume Fraction > Option	Automatic with Value
	Fluid Specific Initialization > Air > Initial Conditions > Volume Fraction > Volume Fraction	0
	Fluid Specific Initialization	Water
	Fluid Specific Initialization > Water > Initial Conditions > Cartesian Velocity Components > Option	Automatic
	Fluid Specific Initialization > Water > Initial Conditions > Cartesian Velocity Components > Velocity Scale	(Selected)
	Fluid Specific Initialization > Water > Initial Conditions > Cartesian Velocity Components > Velocity Scale > Value	0 [m s^-1]

3. Click **OK**.

17.5.6. Setting Solver Control

Generally, two different time scales exist for multiphase mixers. The first is a small time scale based on the rotational speed of the impeller, typically taken as $1/\omega$, resulting in a time scale of 0.11 s for this case. The second time scale is usually larger and based on the recirculation time of the continuous phase in the mixer.

Using a time step based on the rotational speed of the impeller will be more robust, but convergence will be slow since it takes time for the flow field in the mixer to develop. Using a larger time step reduces the number of iterations required for the mixer flow field to develop, but reduces robustness. You will need to experiment to find an optimum time step.

1. Click *Solver Control*  .
2. Apply the following settings:

Tab	Setting	Value
Basic Settings	Advection Scheme > Option	High Resolution
	Convergence Control > Max. Iterations	100[a]
	Convergence Control > Fluid Timescale Control > Timescale Control	Physical Timescale
	Convergence Control > Fluid Timescale Control > Physical Timescale	2 [s][b]
	Convergence Criteria	(Default)[c]

[a]For advice on setting time steps in multiphase simulations, see Timestepping.

[b]This is an aggressive time step for this case.

[c]The default is an RMS value of 1.0E-04. If you are using a maximum edge length of 0.005 m or less to produce a finer mesh, use a target residual of 1.0E-05 to obtain a more accurate solution.

3. Click **OK**.

17.5.7. Adding Monitor Points

You can monitor the value of an expression during the solver run so that you can view the volume fraction of air in the tank (the gas hold up). The gas hold up is often used to judge convergence in these types of simulations by converging until a steady-state value is achieved.

1. Create the following expressions:

```
TankAirHoldUp = volumeAve(Air.vf)@tank
ImpellerAirHoldUp = volumeAve(Air.vf)@impeller
TotalAirHoldUp = (volume()@tank * TankAirHoldUp +
                  volume()@impeller * ImpellerAirHoldUp) /
                  (volume()@tank + volume()@impeller)
```

2. Click *Output Control*  .
3. Apply the following settings:

Tab	Setting	Value
Monitor	Monitor Options	(Selected)

4. Create a new **Monitor Points and Expressions** item named `Total Air Holdup`.
5. Apply the following settings to `Total Air Holdup`:

Setting	Value
Option	Expression
Expression Value	TotalAirHoldUp

6. Click **OK**.

17.5.8. Writing the CFX-Solver Input (.def) File

1. Click *Define Run*  .
2. Apply the following settings:

Setting	Value
File name	MultiphaseMixer.def

3. Click **Save**.

   If you are notified the file already exists, click **Overwrite**.
4. Click **OK**.
5. If using Standalone Mode, quit CFX-Pre, saving the simulation (`.cfx`) file at your discretion.

17.6. Obtaining a Solution Using CFX-Solver Manager

Start the simulation from CFX-Solver Manager:

1. Ensure **Define Run** is displayed.
2. Click **Start Run**.

   CFX-Solver runs and attempts to obtain a solution. This can take a long time depending on your system.
3. Select the check box next to **Post-Process Results** when the completion message appears at the end of the run.
4. If using Standalone Mode, select the check box next to **Shut down CFX-Solver Manager**.
5. Click **OK**.

17.7. Examining the Results in CFD-Post

After CFD-Post has started and the tank and mixer domains have been loaded, the mixer geometry appears in the viewer. Orient the view so that the X-axis points up as follows:

- Right-click a blank area in the viewer and select **Predefined Camera** > **Isometric View (X up)**.

You will create some plots showing the distributions of velocity and other variables. You will also calculate the torque and power required to turn the impeller at 84 rpm.

17.7.1. Creating a Plane Locator

Create a vertical plane that extends from the shaft to the tank wall at a location far from the baffle. This plane will be used as a locator for various plots, such as velocity vector plots and plots showing the distribution of air.

1. When CFD-Post starts, the **Domain Selector** dialog box might appear. If it does, ensure that both the impeller and tank domains are selected, then click **OK** to load the results from these domains.
2. Create a new plane named `Plane 1`.
3. Apply the following settings:

Tab	Setting	Value
Geometry	Definition > Method	Three Points
	Definition > Point 1	1, 0, 0
	Definition > Point 2	0, 1, -0.9
	Definition > Point 3	0, 0, 0

4. Click **Apply**.

17.7.2. Plotting Velocity

Recall that the homogeneous multiphase option was not used when specifying the domain settings (see the setting for **Fluid Models** > **Multiphase Options** > **Homogeneous Model** in *Rotating Domain for the Impeller* (p. 292)). As a consequence, the air and water velocity fields may differ from each other. Plot the velocity of water, then air on `Plane 1`:

1. Create a new vector plot named `Vector 1`.
2. Apply the following settings:

Tab	Setting	Value
Geometry	Definition > Locations	Plane 1
	Variable	Water.Velocity in Stn Frame[a]
Symbol	Symbol Size	0.2
	Normalize Symbols	(Selected)

[a]Using this variable, instead of Water.Velocity, results in the velocity vectors appearing to be continuous at the interface between the rotating and stationary domains. Velocity variables that do not include a frame specification always use the local reference frame.

3. Click **Apply**.
4. Turn off the visibility of `Plane 1` to better see the vector plot.
5. Observe the vector plot (in particular, near the top of the tank). Note that the water is not flowing out of the domain.
6. Change the variable to `Air.Velocity in Stn Frame` and click **Apply**.

   Observe this vector plot, noting how the air moves upward all the way to the water surface, where it escapes.

7. Turn off the visibility of `Vector 1` in preparation for the next plots.

17.7.3. Plotting Pressure Distribution

Color `Plane 1` to see the pressure distribution:

1. Turn on the visibility of `Plane 1`.
2. Apply the following settings to `Plane 1`:

Tab	Setting	Value
Color	Mode	Variable
	Variable	Pressure
	Range	Local

3. Click **Apply**.

   Note that the pressure field computed by the solver excludes the hydrostatic pressure corresponding to the specified buoyancy reference density. The pressure field including this hydrostatic component (as well as the reference pressure) can by visualized by plotting `Absolute Pressure`.

17.7.4. Plotting Volume Fractions

To see the distribution of air, color `Plane 1` by the volume fraction of air:

1. Apply the following settings to `Plane 1`:

Tab	Setting	Value
Color	Mode	Variable
	Variable	Air.Volume Fraction
	Range	User Specified
	Min	0
	Max	0.04

2. Click **Apply**.

The user-specified range was made much narrower than the **Global** and **Local** ranges in order to better show the variation.

17.7.5. Plotting Shear Strain Rate and Shear Stress

Areas of high shear strain rate or shear stress are typically also areas where the highest mixing occurs.

To see where the most of the mixing occurs, color `Plane 1` by shear strain rate.

1. Apply the following settings to `Plane 1`:

Tab	Setting	Value
Color	Variable	Air.Shear Strain Rate
	Range	User Specified

Tab	Setting	Value
	Min	0 [s^-1]
	Max	15 [s^-1]

2. Click **Apply**.

   The user-specified range was made much narrower than the **Global** and **Local** ranges in order to better show the variation.

3. Modify the coloring of the `MultiphaseMixer_001` > `tank` > `tank Default` object by applying the following settings:

Tab	Setting	Value
Color	Mode	Variable
	Variable	Water.Wall Shear
	Range	Local

4. Click **Apply**.

The legend for this plot shows the range of wall shear values.

The global maximum wall shear stress is much higher than the maximum value on the default walls. The global maximum values occur on the `TankShaft` boundary directly above the inlet. Although these values are very high, the shear force exerted on this boundary is small since the contact area fraction of water is very small there.

17.7.6. Calculating Torque and Power Requirements

Calculate the torque and power required to spin the impeller at 84 rpm:

1. Select **Tools** > **Function Calculator** from the main menu or click *Show Function Calculator*  .
2. Apply the following settings:

Tab	Setting	Value
Function Calculator	Function	torque
	Location	Blade Thin Surface Side 1
	Axis	Global X
	Fluid	All Fluids

3. Click **Calculate** to find the torque about the X-axis imparted by both fluids on location `Blade Thin Surface Side 1`.
4. Repeat the calculation, setting **Location** to `Blade Thin Surface Side 2`.

The sum of these two torques is approximately -67.4 [N m] about the X-axis. Multiplying by -4 to find the torque required by all of the impeller blades gives a required torque of approximately 270 [N m] about the X-axis. You could also include the contributions from the locations `HubShaft` and `TankShaft`; however in this case their contributions are negligible.

The power requirement is simply the required torque multiplied by the rotational speed (84 rpm = 8.8 rad/s): Power = 270 N m * 8.8 rad/s = 2376 W.

Remember that this value is the power requirement for the work done on the fluids; it does not account for any mechanical losses, motor efficiencies etc. Also note that the accuracy of these results is significantly affected by the coarseness of the mesh. You should not use a mesh of this length scale to obtain accurate quantitative results.

Chapter 18: Gas-Liquid Flow in an Airlift Reactor

This tutorial includes:

18.1. Tutorial Features

In this tutorial you will learn about:

- Setting up a multiphase flow simulation involving air and water.
- Using a fluid dependent turbulence model to set different turbulence options for each fluid.
- Specifying buoyant flow.
- Specifying a degassing outlet boundary to allow air, but not water, to escape from the boundary.
- Using face culling in CFD-Post to turn off the visibility of one side of a surface.

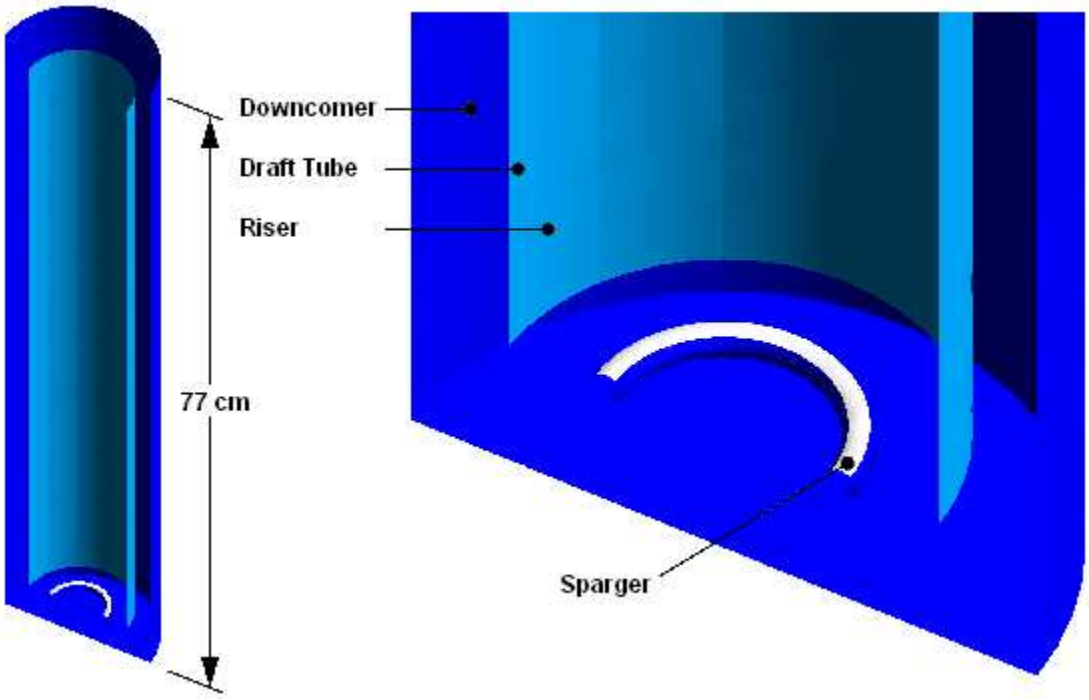
Component	Feature	Details
CFX-Pre	User Mode	General Mode
	Analysis Type	Steady State
	Fluid Type	General Fluid
	Domain Type	Single Domain
	Turbulence Model	Dispersed Phase Zero Equation
		Fluid-Dependent Turbulence Model
		k-Epsilon
	Heat Transfer	None
	Buoyant Flow	
	Multiphase	
	Boundary Conditions	Inlet (Subsonic)
		Outlet (Degassing)
		Symmetry Plane
		Wall: (Slip Depends on Volume Fraction)

Component	Feature	Details
	Timestep	Physical Time Scale
CFD-Post	Plots	Default Locators
		Vector
	Other	Changing the Color Range
		Symmetry

18.2. Overview of the Problem to Solve

This tutorial demonstrates the Eulerian-Eulerian multiphase model in CFX by simulating an airlift reactor. Airlift reactors are tall gas-liquid contacting vessels and are often used in processes where gas absorption is important (for example, bioreactors to dissolve oxygen in broths) and to limit the exposure of micro-organisms to excessive shear imparted by mechanically driven mixers.

Figure 18.1 Cut-away Diagram of the Airlift Reactor



This tutorial models the dispersion of air bubbles in water. Air is supplied through a sparger at the bottom of the vessel and the rising action of the bubbles provides gentle agitation of the water. An internal tube (draft tube) directs recirculation of the flow. The airlift reactor is shown in a cut-away diagram in *Figure 18.1* (p. 310).

Simple airlift reactors that are without a draft tube tend to develop irregular flow patterns and poor overall mixing. The draft tube in the airlift reactor helps to establish a regular flow pattern in the column and to achieve better uniformity in temperature, concentration, and pH in the liquid phase, but sometimes at the expense of decreased mass transfer from gas to liquid.

This tutorial also demonstrates the use of pairs of internal wall boundaries to model thin 3D features. In this case, a pair of wall boundaries is used to model the draft tube. Other applications include baffles and guide vanes. In the postprocessing section of this tutorial, you will learn how to use face culling to hide one side

of a boundary. This technique enables you to independently color each boundary of a pair of back-to-back boundaries (located at the same position in 3D space, but with opposite orientation).

The airlift reactor that is modeled here is very similar to the laboratory bench scale prototype used by García-Calvo and Letón.

A formal analysis of this simulation involving a finer mesh is available at the end of this tutorial. For details, see *Additional Fine Mesh Simulation Results* (p. 322).

18.3. Before You Begin

If this is the first tutorial you are working with, it is important to review the following topics before beginning:

- *Preparing a Working Directory and Starting ANSYS CFX in Standalone Mode* (p. 1)
- *Running ANSYS CFX Tutorials in ANSYS Workbench* (p. 2)
- *Changing the Display Colors* (p. 5)
- *Playing a Tutorial Session File* (p. 4)

18.4. Starting CFX-Pre

1. Prepare the working directory using the following files in the `examples` directory:
   - `BubbleColumn.pre`
   - `BubbleColumnMesh.gtm`
2. Set the working directory and start CFX-Pre.

   For details, see *Preparing a Working Directory and Starting ANSYS CFX in Standalone Mode* (p. 1)

18.5. Defining a Case in CFX-Pre

If you want to set up the simulation automatically and continue to *Obtaining a Solution Using CFX-Solver Manager* (p. 318), run `BubbleColumn.pre`.

1. In CFX-Pre, select **File** > **New Case**.
2. Select **General** and click **OK**.
3. Select **File** > **Save Case As**.
4. Under **File name**, type `BubbleColumn`.
5. Click **Save**.

18.5.1. Importing the Mesh

1. Right-click `Mesh` and select **Import Mesh** > **Other**. The **Import Mesh** dialog box appears.
2. Apply the following settings:

Setting	Value
File name	BubbleColumnMesh.gtm

3. Click **Open**.

18.5.2. Creating the Domain

1. Edit `Case Options` > `General` in the **Outline** tree view and ensure that **Automatic Default Domain** is turned on. A domain named `Default Domain` should now appear under the `Simulation` branch.
2. Double-click **Default Domain**.
3. In the **Basic Settings** tab, under **Fluid and Particle Definitions**, delete `Fluid 1` and create a new fluid definition called `Air`.
4. Use the  button to create a new fluid named `Water`.
5. Apply the following settings:

Tab	Setting	Value
Basic Settings	Location and Type > Location	B1.P3, B2.P3
	Fluid and Particle Definitions	Air
	Fluid and Particle Definitions > Air > Material	Air at 25 C
	Fluid and Particle Definitions > Air > Morphology > Option	Dispersed Fluid
	Fluid and Particle Definitions > Air > Morphology > Mean Diameter	6 [mm]
	Fluid and Particle Definitions	Water
	Fluid and Particle Definitions > Water > Material	Water
	Domain Models > Pressure > Reference Pressure	1 [atm]
	Domain Models > Buoyancy > Option	Buoyant
	Domain Models > Buoyancy > Gravity X Dirn.	0 [m s^-2]
	Domain Models > Buoyancy > Gravity Y Dirn.	-9.81 [m s^-2]
	Domain Models > Buoyancy > Gravity Z Dirn.	0 [m s^-2]
	Domain Models > Buoyancy > Buoy. Ref. Density[a]	997 [kg m^-3]
Fluid Models	Multiphase > Homogeneous Model	(Cleared)[b]
	Multiphase > Free Surface Model > Option	None
	Heat Transfer > Homogeneous Model	(Cleared)
	Heat Transfer > Option	Isothermal
	Heat Transfer > Fluid Temperature	25 [C]
	Turbulence > Homogeneous Model	(Cleared)
	Turbulence > Option	Fluid Dependent[c]
Fluid Pair Models	Fluid Pair	Air | Water
	Fluid Pair > Air | Water > Surface Tension Coefficient	(Selected)
	Fluid Pair > Air | Water > Surface Tension Coefficient > Surf. Tension Coeff.	0.072 [N m^-1][d]
	Fluid Pair > Air | Water > Momentum Transfer > Drag Force > Option	Grace

Tab	Setting	Value
	Fluid Pair > Air | Water > Momentum Transfer > Drag Force > Volume Fraction Correction Exponent	(Selected)
	Fluid Pair > Air | Water > Momentum Transfer > Drag Force > Volume Fraction Correction Exponent > Value	2[e]
	Fluid Pair > Air | Water > Momentum Transfer > Non-drag forces > Turbulent Dispersion Force > Option	Favre Averaged Drag Force
	Fluid Pair > Air | Water > Momentum Transfer > Non-drag forces > Turbulent Dispersion Force > Dispersion Coeff.	1.0
	Fluid Pair > Air | Water > Turbulence Transfer > Option	Sato Enhanced Eddy Viscosity[f]

[a]For dilute dispersed multiphase flow, always set the buoyancy reference density to that for continuous fluid.

[b]Turn off the homogeneous model to allow each fluid to have its own velocity field.

[c]The fluid-specific turbulence settings are defined in the **Fluid Specific Models** tab. They are set to default values.

[d]This must be set to allow the Grace drag model to be used.

[e]A positive value is appropriate for large bubbles. For details, see Densely Distributed Fluid Particles: Grace Drag Model.

[f]This models particle-induced turbulence. For details, see Turbulence Enhancement.

6. Click **OK**.

18.5.3. Creating the Boundary Conditions

For this simulation of the airlift reactor, the required boundary conditions are:

- An inlet for air on the sparger.
- A degassing outlet for air at the liquid surface.
- A pair of wall boundaries for the draft tube.
- An exterior wall for the outer wall, base and sparger tube.
- Symmetry planes.

18.5.3.1. Inlet Boundary

At the sparger, create an inlet boundary that injects air at 0.3 m/s with a volume fraction of 0.25:

1. Create a new boundary named `Sparger`.
2. Apply the following settings:

Tab	Setting	Value
Basic Settings	Boundary Type	Inlet
	Location	Sparger
Boundary Details	Mass And Momentum > Option	Fluid Dependent
Fluid Values	Boundary Conditions	Air
	Boundary Conditions > Air > Velocity > Option	Normal Speed

Tab	Setting	Value
	Boundary Conditions > Air > Velocity > Normal Speed	0.3 [m s^-1]
	Boundary Conditions > Air > Volume Fraction > Option	Value
	Boundary Conditions > Air > Volume Fraction > Volume Fraction	0.25
	Boundary Conditions	Water
	Boundary Conditions > Water > Velocity > Option	Normal Speed
	Boundary Conditions > Water > Velocity > Normal Speed	0 [m s^-1]
	Boundary Conditions > Water > Volume Fraction > Option	Value
	Boundary Conditions > Water > Volume Fraction > Volume Fraction	0.75

3. Click **OK**.

18.5.3.2. Outlet Boundary

Create a degassing outlet boundary at the top of the reactor:

1. Create a new boundary named `Top`.
2. Apply the following settings:

Tab	Setting	Value
Basic Settings	Boundary Type	Outlet
	Location	Top
Boundary Details	Mass And Momentum > Option	Degassing Condition

3. Click **OK**.

18.5.3.3. Draft Tube Boundaries

The draft tube is an infinitely thin surface that requires a wall boundary on both sides; if only one side has a boundary then CFX-Solver will fail.

The `Free Slip` condition can be used for the gas phase since the contact area with the walls is near zero for low gas phase volume fractions.

The required boundary settings are the same for both sides of the draft tube. From the point of view of solving the simulation, you could therefore define a single boundary and choose both sides of the tube as the location. However, the postprocessing section of this tutorial requires the use of separate boundaries in order to illustrate the use of face culling (a visualization technique), so you will create two wall boundaries instead.

Start by creating a wall boundary for the outer side of the draft tube:

1. Create a new boundary named `DraftTube Downcomer Side`.
2. Apply the following settings:

Tab	Setting	Value
Basic Settings	Boundary Type	Wall
	Location	DraftTube
Boundary Details	Mass And Momentum > Option	Fluid Dependent
	Wall Roughness > Option	Smooth Wall
	Wall Contact Model > Option	Use Volume Fraction
Fluid Values	Boundary Conditions	Air
	Boundary Conditions > Air > Mass And Momentum > Option	Free Slip Wall
	Boundary Conditions	Water
	Boundary Conditions > Water > Mass And Momentum > Option	No Slip Wall

3. Click **OK**.

Now create a boundary named `DraftTube Riser Side` using the same settings, but located on `F10.B1.P3` (the riser side of the draft tube).

18.5.3.4. Symmetry Plane Boundary

To simulate the full geometry, create symmetry plane boundary conditions on the `Symmetry1` and `Symmetry2` locators:

1. Create a new boundary named `SymP1`.
2. Apply the following settings:

Tab	Setting	Value
Basic Settings	Boundary Type	Symmetry
	Location	Symmetry1

3. Click **OK**.
4. Create a new boundary named `SymP2`.
5. Apply the following settings:

Tab	Setting	Value
Basic Settings	Boundary Type	Symmetry
	Location	Symmetry2

6. Click **OK**.

18.5.3.5. Modifying the Default Boundary

The remaining external regions are assigned to the default wall boundary. As for the draft tube boundary, set the air phase to use the free slip wall condition:

1. Edit `Default Domain Default`.
2. Apply the following settings:

Tab	Setting	Value
Boundary Details	Mass And Momentum > Option	Fluid Dependent
Fluid Values	Boundary Conditions	Air
	Boundary Conditions > Air > Mass And Momentum > Option	Free Slip Wall
	Boundary Conditions	Water
	Boundary Conditions > Water > Mass And Momentum > Option	No Slip Wall

3. Click **OK**.

The boundary specifications are now complete.

18.5.4. Setting Initial Values

It often helps to set an initial velocity for a dispersed phase that is different to that of the continuous phase. This results in a non-zero drag between the phases which can help stability at the start of a simulation.

For some airlift reactor problems, improved convergence can be obtained by using CEL (CFX Expression Language) to specify a non-zero volume fraction for air in the riser portion and a value of zero in the downcomer portion. This should be done if two solutions are possible (for example, if the flow could go up the downcomer and down the riser).

Set the initial values:

1. Click *Global Initialization*  .

   Since a single pressure field exists for a multiphase calculation, do not set pressure values on a per-fluid basis.

2. Apply the following settings:

Tab	Setting	Value
Fluid Settings	Fluid Specific Initialization	Air
	Fluid Specific Initialization > Air > Initial Conditions > Cartesian Velocity Components > Option	Automatic with Value
	Fluid Specific Initialization > Air > Initial Conditions > Cartesian Velocity Components > U	0 [m s^-1]

Tab	Setting	Value
	Fluid Specific Initialization > Air > Initial Conditions > Cartesian Velocity Components > V	0.3 [m s^-1]
	Fluid Specific Initialization > Air > Initial Conditions > Cartesian Velocity Components > W	0 [m s^-1]
	Fluid Specific Initialization > Air > Initial Conditions > Volume Fraction > Option	Automatic
	Fluid Specific Initialization	Water[a]
	Fluid Specific Initialization > Water > Initial Conditions > Cartesian Velocity Components > Option	Automatic with Value
	Fluid Specific Initialization > Water > Initial Conditions > Cartesian Velocity Components > U	0 [m s^-1]
	Fluid Specific Initialization > Water > Initial Conditions > Cartesian Velocity Components > V	0 [m s^-1]
	Fluid Specific Initialization > Water > Initial Conditions > Cartesian Velocity Components > W	0 [m s^-1]
	Fluid Specific Initialization > Water > Initial Conditions > Volume Fraction > Option	Automatic with Value
	Fluid Specific Initialization > Water > Initial Conditions > Volume Fraction > Volume Fraction	1[b]

[a]Since there is no water entering or leaving the domain, a stationary initial guess is recommended.

[b]The volume fractions must sum to unity over all fluids. Since a value has been set for water, the volume fraction of air will be calculated as the remaining difference, in this case, 0.

3. Click **OK**.

18.5.5. Setting Solver Control

1. Click *Solver Control*  .
2. Apply the following settings:

Tab	Setting	Value
Basic Settings	Advection Scheme > Option	High Resolution
	Convergence Control > Max. Iterations	100[a]
	Convergence Control > Fluid Timescale Control > Timescale Control	Physical Timescale
	Convergence Control > Fluid Timescale Control > Physical Timescale	1 [s]

Tab	Setting	Value
	Convergence Criteria > Residual Target	1.0E-04[b]
	Convergence Criteria > Conservation Target	(Selected)
	Convergence Criteria > Conservation Target > Value	0.01
Advanced Options	Global Dynamic Model Control > Multiphase Control	(Selected)
	Global Dynamic Model Control > Multiphase Control > Volume Fraction Coupling	(Selected)
	Global Dynamic Model Control > Multiphase Control > Volume Fraction Coupling > Option	Coupled [c]

[a]For advice on setting time steps in multiphase simulations, see Timestepping.

[b]If you are using a maximum edge length of `0.005` m or less to produce a finer mesh, use a target residual of 1.0E-05 to obtain a more accurate solution.

[c]Volume Fraction Coupling is recommended when changes in volume fraction have a large effect on momentum transport. This is the case in this tutorial, because it employs a model for the turbulent dispersion in which the force is proportional to the volume fraction gradient. Other cases where Volume Fraction Coupling is recommended include those in which solid pressure forces are used and those in which gravitational forces are significant (for example, free surface flows).

3. Click **OK**.

18.5.6. Writing the CFX-Solver Input (.def) File

1. Click *Define Run*  .
2. Apply the following settings:

Setting	Value
File name	BubbleColumn.def

3. Click **Save**.

   CFX-Solver Manager automatically starts and, on the **Define Run** dialog box, the **Solver Input File** is set.
4. If using Standalone Mode, quit CFX-Pre, saving the simulation (`.cfx`) file at your discretion.

18.6. Obtaining a Solution Using CFX-Solver Manager

Start the simulation from CFX-Solver Manager:

Note

If you are using a fine mesh for a formal quantitative analysis of the flow in the reactor, the solution time will be significantly longer than for the coarse mesh. You can run the simulation in parallel to reduce the solution time. For details, see *Obtaining a Solution in Parallel* (p. 118).

1. Ensure **Define Run** is displayed.
2. Click **Start Run**.

   CFX-Solver runs and attempts to obtain a solution. This can take a long time depending on your system.
3. Select the check box next to **Post-Process Results** when the completion message appears at the end of the run.
4. If using Standalone Mode, select the check box next to **Shut down CFX-Solver Manager**.
5. Click **OK**.

18.7. Viewing the Results in CFD-Post

You will first create plots of velocity and volume fraction. You will then display the entire geometry.

Because the simulation in this tutorial is conducted on a coarse grid, the results are only suitable for a qualitative demonstration of the multiphase capability of ANSYS CFX.

The following topics will be discussed:

18.7.1. Creating Water Velocity Vector Plots

1. Right-click a blank area in the viewer and select **Predefined Camera** > **View From -Z**.
2. Create a new vector plot named `Vector 1`.
3. Apply the following settings:

Tab	Setting	Value
Geometry	Definition > Locations	SymP1
	Definition > Variable	Water.Velocity
Color	Range	User Specified
	Min	0 [m s^-1]
	Max	1 [m s^-1]
Symbol	Symbol Size	0.3

4. Click **Apply**.
5. In the tree view, right-click `Vector 1`, select **Duplicate**, and click **OK** to accept the default name, `Vector 2`.
6. Edit `Vector 2`.
7. On the **Geometry** tab, set **Definition** > **Variable** to `Air.Velocity` and click **Apply**.
8. Compare `Vector 1` and `Vector 2` by toggling the visibility of each one.

   Zoom in as required.

   Observe that the air flows upward, leaving the tank, at the degassing outlet.

Note that the air rises faster than the water in the riser and descends slower than the water in the downcomer.

18.7.2. Creating Volume Fraction Plots

Plot the volume fraction of air on the symmetry plane SymP1:

1. Right-click a blank area in the viewer and select **Predefined Camera** > **View From -Z**.
2. Turn on the visibility of SymP1.
3. Edit SymP1.
4. Apply the following settings to SymP1:

Tab	Setting	Value
Color	Mode	Variable
	Variable	Air.Volume Fraction
	Range	User Specified
	Min	0
	Max	0.025

5. Click **Apply**.

   Observe the volume fraction values throughout the domain.
6. Turn off the visibility of SymP1.

Next, plot the volume fraction of air on each side of the draft tube:

1. Right-click a blank area in the viewer and select **Predefined Camera** > **Isometric View (Y up)**.
2. Turn on the visibility of DraftTube Downcomer Side.
3. Modify DraftTube Downcomer Side by applying the following settings:

Tab	Setting	Value
Color	Mode	Variable
	Variable	Air.Volume Fraction
	Range	User Specified
	Min	0
	Max	0.025

4. Click **Apply**.
5. Rotate the plot in the viewer to see both sides of DraftTube Downcomer Side.

   Notice that the plot appears on both sides of the DraftTube Downcomer Side boundary. When viewing plots on internal surfaces, you must ensure that you are viewing the correct side. You will make use of the face culling rendering feature to turn off the visibility of the plot on the side of the boundary for which the plot does not apply.

   The DraftTube Downcomer Side boundary represents the side of the internal surface in the downcomer (in this case, outer) region of the reactor. To confirm this, you could make a vector plot

of the variable `Normal` (representing the face normal vectors) on the locator `DraftTube Downcomer Side`; the fluid is on the side opposite the normal vectors.

In this case, you need to turn off the "front" faces of the plot; The front faces are, by definition, on the same side of the plot as the normal vectors.

6. Modify `DraftTube Downcomer Side` by applying the following settings:

Tab	Setting	Value
Render	Show Faces > Face Culling	Front Faces

7. Click **Apply**.
8. Rotate the image in the viewer to see the effect of face culling on `DraftTube Downcomer Side`. You should see that the color appears on the downcomer side only.
9. Turn on the visibility of `DraftTube Riser Side`.
10. Color the `DraftTube Riser Side` object using the same color and rendering settings as for `DraftTube Downcomer Side`.

    The normal vectors for `DraftTube Riser Side` point opposite to those of `DraftTube Downcomer Side`, so the faces on `DraftTube Riser Side` are plotted only on the riser (in this case, inner) side of the airlift reactor.

Rotating the geometry will now correctly show the air volume fraction on each side of the draft tube.

Face culling was needed to prevent interference between the plots on each side of the draft tube. To demonstrate this, try turning off face culling for `DraftTube Downcomer Side` and watch the effect on the riser side. You might notice that the plot from the downcomer side interferes with, or even completely overrides, the riser-side plot. Results may vary, which is why face culling should always be used to prevent possible interference.

18.7.3. Displaying the Entire Airlift Reactor Geometry

Display the entire airlift reactor geometry by expanding `User Locations and Plots` and double-clicking the `Default Transform` object:

1. Apply the following settings to `Default Transform`:

Tab	Setting	Value
Definition	Instancing Info From Domain	(Cleared)
	# of Copies	12
	Apply Rotation	(Selected)
	Apply Rotation > Method	Principal Axis
	Apply Rotation > Axis	Y
	Apply Rotation > # of Passages	12

2. Click **Apply**.

18.8. Additional Fine Mesh Simulation Results

A formal analysis of this airlift reactor was carried out on a finer grid (having 21000+ nodes and a maximum edge length of 0.005 m).

The analysis showed a region of air bubble recirculation at the top of the reactor on the downcomer side. This was confirmed by zooming in on a vector plot of Air.Velocity on SymP1 near the top of the downcomer. A similar plot of Water.Velocity revealed no recirculation of the water.

Other results of the simulation:

- Due to their large 0.006 m diameter, the air bubbles quickly attained a significant terminal slip velocity (i.e., the terminal velocity relative to water). The resulting terminal slip velocity, obtained using the Grace drag model, is consistent with the prediction by Maneri and Mendelson and the prediction by Baker and Chao. These correlations predict a terminal slip velocity of about 0.23 m s^{-1} to 0.25 m s^{-1} for air bubbles of the diameter specified.
- The values of gas hold-up (the average volume fraction of air in the riser), the superficial gas velocity (the rising velocity, relative to the reactor vessel, of gas bubbles in the riser, multiplied by the gas hold-up), and the liquid velocity in the downcomer agree with the results reported by García-Calvo and León, for gas hold-up values of 0.03 or less. At higher values of gas hold-up, the multifluid model does not account for pressure-volume work transferred from gas to liquid due to isothermal expansion of the bubbles. The simulation therefore tends to under-predict both the superficial gas velocity in the riser, and the liquid velocity in the downcomer for gas hold-up values greater than 0.03.

Note

Multiphase results files contain the vector variable Fluid.Superficial Velocity defined as Fluid.Volume Fraction multiplied by Fluid.Velocity. This is sometimes also referred to as the fluid volume flux. The components of this vector variable are available as scalar variables (for example, Fluid.Superficial Velocity X).

Many reference texts on airlift reactors cite the Hughmark correlation as a standard for gas hold-up and superficial gas velocity in airlift reactors. However, the Hughmark correlation should not be used when liquid flow is concurrent with gas at velocities exceeding 0.1 m/s. In the airlift reactor described in this tutorial, the liquid velocity in the riser clearly exceeds 0.2 m/s, and the Hughmark correlation is therefore not applicable.

Chapter 19: Air Conditioning Simulation

This tutorial includes:

Important

You must have the required Fortran compiler installed and set in your system path in order to run this tutorial. For details on which Fortran compiler is required for your platform, see the applicable ANSYS, Inc. installation guide. If you are not sure which Fortran compiler is installed on your system, try running the `cfx5mkext` command (found in `<CFXROOT>/bin`) from the command line and read the output messages.

Note

This tutorial requires that you run the CFX-Solver in serial mode (not parallel mode).

19.1. Tutorial Features

In this tutorial you will learn about:

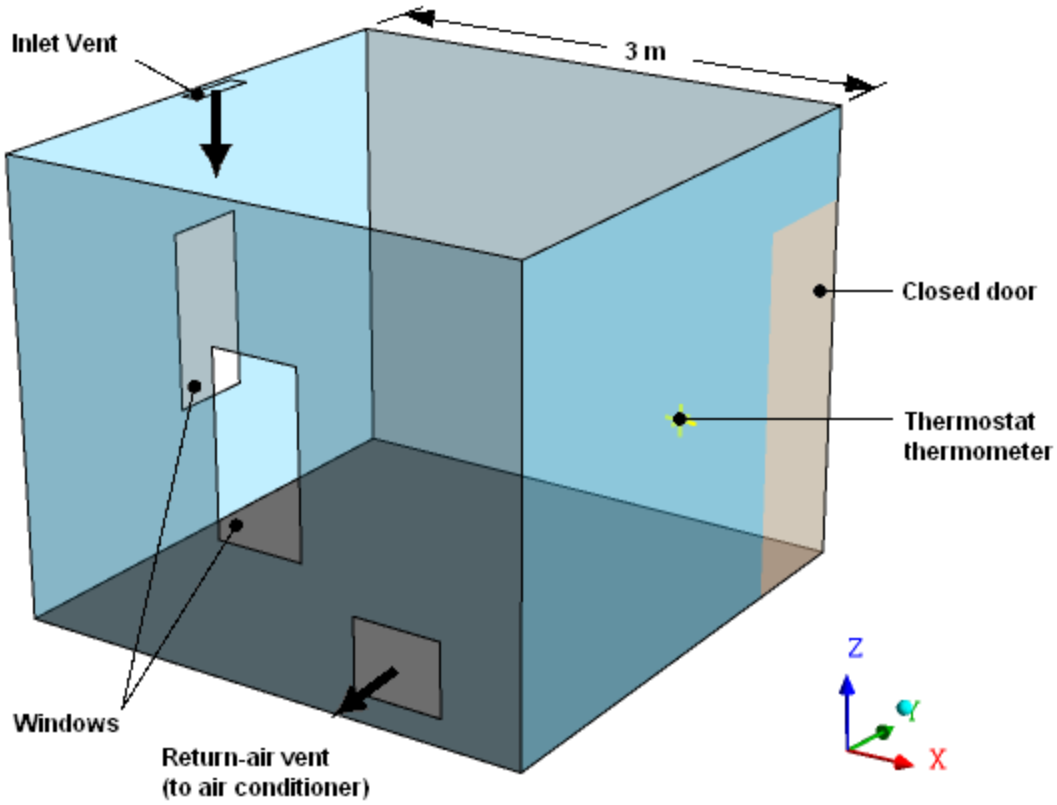
- Importing CEL expressions.
- @REGION CEL syntax.
- Setting up a user CEL function.
- Setting up a monitor point to observe the temperature at a prescribed location.
- Using the Monte Carlo radiation model with a directional source of radiation.
- Post-processing a transient simulation.

Component	Feature	Details
CFX-Pre	User Mode	General Mode
	Analysis Type	Transient
	Fluid Type	General Fluid
	Domain Type	Single Domain
	Turbulence Model	k-Epsilon

Component	Feature	Details
	Heat Transfer	Thermal Energy
	Radiation	Monte Carlo
	Buoyant Flow	
	Boundary Conditions	Boundary Profile Visualization
		Inlet (Profile)
		Outlet (Subsonic)
		Wall: No-Slip
		Wall: Adiabatic
		Wall: Fixed Temperature
	Output Control	Transient Results Files
		Monitor Points
	CEL (CFX Expression Language)	User CEL Function
CFD-Post	Plots	Animation
		Isosurface
		Point
		Slice Plane
	Other	Text Label with Auto Annotation
		Changing the Color Range
		Legend
		Time Step Selection
		Transient Animation with Movie Generation

19.2. Overview of the Problem to Solve

This tutorial simulates a room with a thermostat-controlled air conditioner.



The thermostat switches the air conditioner on and off based on the following data:

- A set point of 22°C (the temperature at or above which the air conditioner turns on)
- A temperature tolerance of 2°C (the amount by which cooling continues below the set point before the air conditioner turns off)
- The temperature at a wall-mounted thermometer

Air flows in steadily from an inlet vent on the ceiling, and flows out through a return-air vent near the floor. When the air conditioner is turned on, the incoming air temperature is reduced compared to the outgoing air temperature. When the air conditioner is turned off, the incoming air temperature is set equal to the outgoing air temperature.

Two windows allow sunlight to enter and heat the room. The walls (including a closed door) and windows are assumed to be at a constant 26°C. The simulation is transient, and continues long enough to allow the air conditioner to cycle on and off.

19.3. Before You Begin

If this is the first tutorial you are working with, it is important to review the following topics before beginning:

- *Preparing a Working Directory and Starting ANSYS CFX in Standalone Mode* (p. 1)
- *Running ANSYS CFX Tutorials in ANSYS Workbench* (p. 2)
- *Changing the Display Colors* (p. 5)
- *Playing a Tutorial Session File* (p. 4)

19.4. Starting CFX-Pre

1. Prepare the working directory using the following files in the `examples` directory:
   - `HVAC.pre`
   - `HVAC_expressions.ccl`
   - `HVACMesh.gtm`
   - `TStat_Control.F`

   Note

   You must have a Fortran compiler installed on your system to perform this tutorial.

2. Set the working directory and start CFX-Pre.

   For details, see *Preparing a Working Directory and Starting ANSYS CFX in Standalone Mode* (p. 1)

19.5. Defining a Case in CFX-Pre

If you want to set up the simulation automatically using a tutorial session file, run `HVAC.pre`. For details, see *Playing a Tutorial Session File* (p. 4). Then proceed to *Obtaining a Solution Using CFX-Solver Manager* (p. 337).

If you want to set up the simulation manually, proceed to the following steps:

1. In CFX-Pre, select **File** > **New Case**.
2. Select **General** and click **OK**.
3. Select **File** > **Save Case As**.
4. Under **File name**, type HVAC.
5. Click **Save**.

19.5.1. Importing the Mesh

1. Right-click `Mesh` and select **Import Mesh** > **CFX Mesh**.

   The **Import Mesh** dialog box appears.

2. Apply the following settings:

Setting	Value
File name	HVACMesh.gtm

3. Click **Open**.
4. Right-click a blank area in the viewer and select **Predefined Camera** > **Isometric View (Z up)** from the shortcut menu.

19.5.2. Importing CEL Expressions

This tutorial uses several CEL expressions to store parameters and to evaluate other quantities that are required by the simulation. Import all of the expressions from the provided file:

1. Select **File** > **Import** > **CCL**.
2. Ensure that **Import Method** is set to **Append**.
3. Select `HVAC_expressions.ccl`, which should be in your working directory.
4. Click **Open**.

The table below lists the expressions, along with the definition and information for each expression:

Expression Name	Expression Definition	Information
ACOn	Thermostat Function(TSensor,TSet,TTol,aitern)	On/off status of the air conditioner (determined by calling a user CEL function with the thermometer temperature, thermostat set point, and temperature tolerance).
Cool TempCalc	TVentOut - (HeatRemoved / (MassFlow * 1.004 [kJ kg^-1 K^-1]))	Temperature of air at the return-air vent (determined by a CEL function).
Flowrate	0.06 [m^3 s^-1]	Volumetric flow rate of air entering the room.
HeatRemoved	1000 [J s^-1]	Rate of thermal energy removal when the air conditioner is on.
MassFlow	1.185 [kg m^-3] * Flowrate	Mass flow rate of air entering the room.
TIn	ACOn*CoolTempCalc+(1-ACOn)*TVentOut	Temperature of inlet vent air (a function of the air conditioner on/off status, return-air vent temperature, thermal energy removal rate, and mass flow rate).
TSensor	probe(T)@Thermometer	Thermometer temperature (determined by a CEL function that gets temperature data from a monitor point).
TSet	22 [C]	Thermometer set point.
TTol	2 [K]	Temperature tolerance.
TVentOut	areaAve(T)@REGION:VentOut	Temperature of outlet vent air.
XCompInlet	5*(x-0.05 [m]) / 1 [m]	Direction vector components for guiding the inlet vent air in a diverging manner as it enters the room.
ZCompInlet	-1+XCompInlet	
tStep	3 [s]	Time step size.
tTotal	225 [s]	Total time.

The CEL function that evaluates the thermometer temperature relies on a monitor point that you will create later in this tutorial.

The CEL expression for the air conditioner on/off status requires a compiled Fortran subroutine and a user CEL function that uses the subroutine. These are created next, starting with the compiled subroutine.

Note

The expression for the return-air vent temperature, TVentOut, makes use of @REGION CEL syntax, which indicates the mesh region named VentOut, rather than a boundary named VentOut. For details about @REGION CEL syntax, see Using Locators in Expressions in the *CFX Reference Guide*.

19.5.3. Compiling the Fortran Subroutine for the Thermostat

A Fortran subroutine that simulates the thermostat is provided in the installation directory for your software (<CFXROOT>/examples/). Before the subroutine can be used, it must be compiled for your platform.

You can compile the subroutine at any time before running CFX-Solver. The operation is performed at this point in the tutorial so that you have a better understanding of the values you need to specify in CFX-Pre when creating a User CEL Function. The cfx5mkext command is used to compile the subroutine as described below.

1. Copy the subroutine TStat_Control.F to your working directory (if you have not already done so).
2. Examine the contents of this file in any text editor to gain a better understanding of this subroutine.

   This file was created by modifying the ucf_template.F file, which is available in the <CFXROOT>/examples/ directory.
3. Select **Tools** > **Command Editor**.
4. Type the following command in the **Command Editor** dialog box (make sure you do not miss the semicolon at the end of the line):

   ```
   ! system ("cfx5mkext TStat_Control.F") == 0 or die "cfx5mkext failed";
   ```

   - This is equivalent to executing the following at a command prompt:

     ```
     cfx5mkext TStat_Control.F
     ```

   - The ! indicates that the following line is to be interpreted as power syntax and not CCL. Everything after the ! symbol is processed as Perl commands.
   - system is a Perl function to execute a system command.
   - The "== 0 or die" will cause an error message to be returned if, for some reason, there is an error in processing the command.
5. Click **Process** to compile the subroutine.

Note

You can use the -double option to compile the subroutine for use with double precision CFX-Solver executables. That is:

```
cfx5mkext -double TStat_Control.F
```

A subdirectory will have been created in your working directory whose name is system dependent (for example, on Linux it is named linux). This subdirectory contains the shared object library.

Note

If you are running problems in parallel over multiple platforms then you will need to create these subdirectories using the `cfx5mkext` command for each different platform.

- You can view more details about the `cfx5mkext` command by running:

```
cfx5mkext -help
```

- You can set a Library Name and Library Path using the `-name` and `-dest` options respectively.
- If these are not specified, the default Library Name is that of your Fortran file and the default Library Path is your current working directory.
- Close the **Command Editor** dialog box.

19.5.4. Creating a User CEL Function for the Thermostat

The expression for the air conditioner on/off status is named ACOn. This expression consists of a call to a user CEL function, `Thermostat Function`, which needs to be created.

Before you create the user CEL function, you will first create a user routine that holds basic information about the compiled Fortran subroutine. You will then create a user function, `Thermostat Function`, so that it is associated with the user routine.

Create the user routine:

1. From the main menu, select **Insert** > **Expressions, Functions and Variables** > **User Routine** or click *User Routine*  .
2. Set the name to `Thermostat Routine`.
3. Apply the following settings:

Tab	Setting	Value
Basic Settings	Option	User CEL Function
	Calling Name	ac_on[a]
	Library Name	TStat_Control[b]
	Library Path	(Working Directory)[c]

[a]This is the name of the subroutine within the Fortran file. Always use lower case letters for the calling name, even if the subroutine name in the Fortran file is in upper case.

[b]This is the name passed to the `cfx5mkext` command by the -name option. If the -name option is not specified, a default is used. The default is the Fortran file name without the `.F` extension.

[c]Set this to your working directory.

4. Click **OK**.

Create the user CEL function:

1. From the main menu, select **Insert** > **Expressions, Functions and Variables** > **User Function** or click *User Function*  .
2. Set the name to `Thermostat Function`.

3. Apply the following settings:

Tab	Setting	Value
Basic Settings	Option	User Function
	User Routine Name	Thermostat Routine
	Argument Units	[K], [K], [K], []a
	Result Units	[]b

[a]These are the units for the four input arguments: TSensor, TSet, TTol, and aitern.

[b]The result will be a dimensionless flag with a value of 0 or 1.

4. Click **OK**.

19.5.5. Setting the Analysis Type

This is a transient simulation. The total time and time step are specified by expressions that you imported earlier. Use these expressions to set up the `Analysis Type` information:

1. Click *Analysis Type*  .
2. Apply the following settings:

Tab	Setting	Value
Basic Settings	Analysis Type > Option	Transient
	Analysis Type > Time Duration > Option	Total Time
	Analysis Type > Time Duration > Total Time	tTotal
	Analysis Type > Time Steps > Option	Timesteps
	Analysis Type > Time Steps > Timesteps	tStep
	Analysis Type > Initial Time > Option	Automatic with Value
	Analysis Type > Initial Time > Time	0 [s]

3. Click **OK**.

19.5.6. Creating the Domain

The domain that models the room air should model buoyancy, given the expected temperature differences, air speeds, and the size and geometry of the room. The domain must model radiation, since directional radiation (representing sunlight) will be emitted from the windows.

Create the domain:

1. Edit `Case Options` > `General` in the **Outline** tree view and ensure that **Automatic Default Domain** is turned on.

A domain named `Default Domain` should appear under the `Simulation` branch.

2. Edit `Default Domain` and apply the following settings:

Tab	Setting	Value
Basic Settings	Location and Type > Location	B1.P3
	Fluid and Particle Definitions	Fluid 1
	Fluid and Particle Definitions > Fluid 1 > Material	Air Ideal Gas
	Domain Models > Pressure > Reference Pressure	1 [atm]
	Domain Models > Buoyancy > Option	Buoyant
	Domain Models > Buoyancy > Gravity X Dirn.	0 [m s^-2]
	Domain Models > Buoyancy > Gravity Y Dirn.	0 [m s^-2]
	Domain Models > Buoyancy > Gravity Z Dirn.	-g
	Domain Models > Buoyancy > Buoy. Ref. Density	1.2 [kg m^-3]
Fluid Models	Heat Transfer > Option	Thermal Energy
	Thermal Radiation > Option	Monte Carlo

3. Click **OK**.

19.5.7. Creating the Boundaries

In this section you will define several boundaries:

- An inlet vent that injects air into the room.
- A return-air vent that lets air leave the room.
- Fixed-temperature windows that emit directed radiation.
- Fixed-temperature walls.

19.5.7.1. Inlet Boundary

Create a boundary for the inlet vent, using the previously-loaded expressions for mass flow rate, flow direction, and temperature:

1. Create a boundary named `Inlet`.
2. Apply the following settings:

Tab	Setting	Value
Basic Settings	Boundary Type	Inlet
	Location	Inlet
Boundary Details	Mass and Momentum > Option	Mass Flow Rate

Tab	Setting	Value
	Mass and Momentum > Mass Flow Rate	MassFlow
	Flow Direction > Option	Cartesian Components
	Flow Direction > X Component	XCompInlet
	Flow Direction > Y Component	0
	Flow Direction > Z Component	ZCompInlet
	Heat Transfer > Option	Static Temperature
	Heat Transfer > Static Temperature	TIn
Plot Options	Boundary Vector	(Selected)
	Boundary Vector > Profile Vec. Comps.	Cartesian Components

3. Click **OK**.

   The viewer shows the inlet velocity profile applied at the inlet, which uses the expressions `XCompInlet` and `ZCompInlet` to specify a diverging flow pattern.

Note

Ignore the physics errors that appear. They will be fixed by setting up the rest of the simulation. The error concerning the expression `TIn` is due to a reference to `Thermometer` which does not yet exist. A monitor point named `Thermometer` will be created later as part of the output control settings.

19.5.7.2. Outlet Boundary

Create a boundary for the return-air vent, specifying a relative pressure of 0 Pa:

1. Create a boundary named `VentOut`.
2. Apply the following settings:

Tab	Setting	Value
Basic Settings	Boundary Type	Outlet
	Location	VentOut
Boundary Details	Mass and Momentum > Option	Average Static Pressure
	Mass and Momentum > Relative Pressure	0 [Pa]

3. Click **OK**.

19.5.7.3. Window Boundary

To model sunlight entering the room, the windows are required to emit directional radiation. To approximate the effect of the outdoor air temperature, assume that the windows have a fixed temperature of 26°C.

Create a boundary for the windows using a fixed temperature of 26°C and apply a radiation source of 600 W m^-2 in the (1, 1, -1) direction:

1. Create a boundary named `Windows`.
2. Apply the following settings:

Tab	Setting	Value
Basic Settings	Boundary Type	Wall
	Location	Window1,Window2
Boundary Details	Heat Transfer > Option	Temperature
	Heat Transfer > Fixed Temperature	26 [C]
Sources	Boundary Source	(Selected)
	Boundary Source > Sources	(Selected)

3. Create a new radiation source item by clicking *Add new item*  and accepting the default name.
4. Apply the following settings to `Radiation Source 1`:

Setting	Value
Option	Directional Radiation Flux
Radiation Flux	600 [W m^-2]
Direction > Option	Cartesian Components
Direction > X Component	1
Direction > Y Component	1
Direction > Z Component	-1

5. Apply the following settings:

Tab	Setting	Value
Plot Options	Boundary Vector	(Selected)
	Boundary Vector > Profile Vec. Comps.	Cartesian Components in Radiation Source 1

6. Click **OK**.

   The direction of the radiation is shown in the viewer.

19.5.7.4. Default Wall Boundary

The default boundary for any undefined surface in CFX-Pre is a no-slip, smooth, adiabatic wall. For this simulation, assume that the walls have a fixed temperature of 26°C. A more detailed simulation would model heat transfer through the walls.

For radiation purposes, assume that the default wall is a perfectly absorbing and emitting surface (emissivity = 1).

Set up the default wall boundary:

1. Edit the boundary named `Default Domain Default`.
2. Apply the following settings:

Tab	Setting	Value
Boundary Details	Heat Transfer > Option	Temperature
	Heat Transfer > Fixed Temperature	26 [C]
	Thermal Radiation > Option	Opaque
	Thermal Radiation > Emissivity	1

3. Click **OK**.

Because this boundary is opaque with an emissivity of 1, all of the radiation is absorbed and none of the radiation is reflected. With no reflected radiation, the **Diffuse Fraction** setting has no effect. For lower values of emissivity, some radiation is reflected, and the **Diffuse Fraction** setting controls the fraction of the reflected radiation that is diffuse, with the remainder being specular (directional).

The default wall boundary includes the `Door` region, which is modeled as a wall (closed door) for simplicity. Since the `Door` region is part of the entire default boundary, it will not appear in the `Wireframe` object that appears in the viewer when the results file of the simulation is opened in CFD-Post, but it can still be viewed as a mesh region.

19.5.8. Setting Initial Values

1. Click *Global Initialization*  .
2. Apply the following settings:

Tab	Setting	Value
Global Settings	Initial Conditions > Velocity Type	Cartesian
	Initial Conditions > Cartesian Velocity Components > Option	Automatic with Value
	Initial Conditions > Cartesian Velocity Components > U	0 [m s^-1]
	Initial Conditions > Cartesian Velocity Components > V	0 [m s^-1]
	Initial Conditions > Cartesian Velocity Components > W	0 [m s^-1]
	Initial Conditions > Static Pressure > Option	Automatic with Value
	Initial Conditions > Static Pressure > Relative Pressure	0 [Pa]
	Initial Conditions > Temperature > Option	Automatic with Value
	Initial Conditions > Temperature > Temperature	22 [C]
	Initial Conditions > Turbulence > Option	Intensity and Length Scale

Tab	Setting	Value
	Initial Conditions > Turbulence > Fractional Intensity > Option	Automatic with Value
	Initial Conditions > Turbulence > Fractional Intensity > Value	0.05
	Initial Conditions > Turbulence > Eddy Length Scale > Option	Automatic with Value
	Initial Conditions > Turbulence > Eddy Length Scale > Value	0.25 [m]
	Initial Conditions > Radiation Intensity > Option	Automatic with Value
	Initial Conditions > Radiation Intensity > Blackbody Temperature	(Selected)
	Initial Conditions > Radiation Intensity > Blackbody Temperature > Blackbody Temp.	22 [C][a]

[a]The initial blackbody temperature of the air should be set to the initial temperature of the air.

3. Click **OK**.

19.5.9. Setting Solver Control

In a typical transient simulation, there should be sufficient coefficient loops per time step to achieve convergence. In order to reduce the time required to run this particular simulation, reduce the maximum number of coefficient loops per time step to 3:

1. Click *Solver Control*  .
2. Apply the following setting:

Tab	Setting	Value
Basic Settings	Convergence Control > Max. Coeff. Loops	3

3. Click **OK**.

19.5.10. Setting Output Control

Set up the solver to output transient results files that record pressure, radiation intensity, temperature, and velocity, on every time step:

1. Click *Output Control*  .
2. Click the **Trn Results** tab.
3. In the **Transient Results** list box, click *Add new item*  , set **Name** to `Transient Results 1`, and click **OK**.
4. Apply the following settings to `Transient Results 1`:

Setting	Value
Option	Selected Variables
Output Variables List[a]	Pressure, Radiation Intensity, Temperature, Velocity
Output Variable Operators	(Selected)
Output Variable Operators > Output Var. Operators	All[b]
Output Frequency > Option	Every Timestep

[a]Use the **Ctrl** key to select more than one variable.

[b]This causes the gradients of the selected variables to be written to the transient results files, along with other information.

To create the thermostat thermometer, set up a monitor point at the thermometer location. Also set up monitors to track the expressions for the temperature at the inlet, the temperature at the outlet, and the on/off status of the air conditioner.

1. Apply the following settings:

Tab	Setting	Value
Monitor	Monitor Objects	(Selected)

2. Create a new **Monitor Points and Expressions** item named `Thermometer`.
3. Apply the following settings to `Thermometer`:

Setting	Value
Output Variables List	Temperature
Cartesian Coordinates	2.95, 1.5, 1.25

4. Create a new **Monitor Points and Expressions** item named `Temp at Inlet`.
5. Apply the following settings to `Temp at Inlet`:

Setting	Value
Option	Expression
Expression Value	TIn

6. Create a new **Monitor Points and Expressions** item named `Temp at VentOut`.
7. Apply the following settings to `Temp at VentOut`:

Setting	Value
Option	Expression
Expression Value	TVentOut

8. Create a new **Monitor Points and Expressions** item named `ACOnStatus`.
9. Apply the following settings to `ACOnStatus`:

Setting	Value
Option	Expression
Expression Value	ACOn

10. Click **OK**.

19.5.11. Writing the CFX-Solver Input (.def) File

1. Click *Define Run*  .
2. Apply the following settings:

Setting	Value
File name	HVAC.def

3. Click **Save**.

   CFX-Solver Manager automatically starts and, on the **Define Run** dialog box, the **Solver Input File** is set.
4. If using Standalone Mode, quit CFX-Pre, saving the simulation (`.cfx`) file at your discretion.

19.6. Obtaining a Solution Using CFX-Solver Manager

When CFX-Pre has shut down and CFX-Solver Manager has started, start the solver and view the monitor points as the solution progresses:

1. Click **Start Run**.

   After a few minutes, a **User Points** tab will appear.

   On that tab, plots will appear showing the values of the monitor points:

   - `ACOnStatus`
   - `Temp at Inlet`
   - `Temp at VentOut`
   - `Thermometer (Temperature)`

2. Click one of the plot lines to see the value of the plot variable at that point.
3. It is difficult to see the plot values because all of the monitor points are plotted on the same scale. To see the plots in more detail, try displaying subsets of them as follows:

   1. Right-click in the plot area and select **Monitor Properties** from the shortcut menu.
   2. In the **Monitor Properties** dialog box, on the **Plot Lines** tab, expand the USER `POINT` branch in the tree.
   3. Clear the check boxes beside all of the monitor points except `ACOnStatus`, then click **Apply**.
   4. Observe the plot for `ACOnStatus`.

      You might have to move the dialog box out of the way to see the plot.

5. In the **Monitor Properties** dialog box, toggle each of the check boxes beside the monitor points, so that all of the monitor points are selected except for `ACOnStatus`, then click **Apply**.
6. Observe the plots for `Temp at Inlet`, `Temp at VentOut`, and `Thermometer (Temperature)`.
7. Click **OK** to close the **Monitor Properties** dialog box.

4. Select the check box next to **Post-Process Results** when the completion message appears at the end of the run.
5. If using Standalone Mode, select the check box next to **Shut down CFX-Solver Manager**.
6. Click **OK**.

19.7. Viewing the Results in CFD-Post

You will first create some graphic objects to visualize the temperature distribution and the thermometer location. You will then create an animation to show how the temperature distribution changes.

19.7.1. Creating Graphics Objects

In this section, you will create two planes and an isosurface of constant temperature, all colored by temperature. You will also create a color legend and a text label that reports the thermometer temperature.

19.7.1.1. Creating Planes

In order to show the key features of the temperature distribution, create two planes colored by temperature as follows:

1. Load the res file (`HVAC_001.res`) if you did not elect to load the results directly from CFX-Solver Manager.
2. Right-click a blank area in the viewer, select **Predefined Camera** > **Isometric View (Z up)**.
3. Create a ZX-Plane named `Plane 1` with **Y**=`1.5 [m]`. Color it by `Temperature` using a user specified range from `19 [C]` to `23 [C]`. Turn off lighting (on the **Render** tab) so that the colors are accurate and can be interpreted properly using the legend.
4. Create an XY plane named `Plane 2` with **Z**=`0.35 [m]`. Color it using the same settings as for the first plane, and turn off lighting.

19.7.1.2. Creating an Isosurface

In order to show the plumes of cool air from the inlet vent, create a surface of constant temperature as follows:

1. Click *Timestep Selector*  . The **Timestep Selector** dialog box appears.
2. Double-click the value: 12 s.

   The time step is set to 12 s so that the cold air plume is visible.
3. Create an isosurface named `Cold Plume` as a surface of `Temperature` = 19 °C.
4. Color the isosurface by `Temperature` (select `Use Plot Variable`) and use the same color range as for the planes. Although the color of the isosurface will not show variation (by definition), it will be consistent with the coloration of the planes.

5. On the **Render** tab for the isosurface, set **Transparency** to `0.5`. Leave lighting turned on to help show the 3D shape of the isosurface.
6. Click **Apply**.

Note

The isosurface will not be visible in some time steps, but you will be able to see it when playing the animation (a step carried out later).

19.7.1.3. Adjusting the Legend

The legend title should not name the locator of any particular object since all objects are colored by the same variable and use the same range. Remove the locator name from the title and, in preparation for making an MPEG video later in this tutorial, increase the text size:

1. Edit `Default Legend View 1`.
2. On the **Definition** tab, change **Title Mode** to `Variable`.

   This will remove the locator name from the legend.
3. Click the **Appearance** tab, then:
   1. Change **Precision** to 2, `Fixed`.
   2. Change **Text Height** to `0.03`.
4. Click **Apply**.

19.7.1.4. Creating a Point for the Thermometer

In the next section, you will create a text label that displays the value of the expression `TSensor`, which represents the thermometer temperature. During the solver run, this expression was evaluated using a monitor point named `Thermometer`. Although this monitor point data is stored in the results file, it cannot be accessed. In order to support the expression for `TSensor`, create a point called `Thermometer` at the same location:

1. From the main menu, select **Insert** > **Location** > **Point**.
2. Set **Name** to `Thermometer`.
3. Set **Point** to `(2.95,1.5,1.25)`.
4. Click **Apply**.

   A marker appears at the thermometer location in the viewer.

19.7.1.5. Creating a Text Label

Create a text label that shows the currently-selected time step and thermometer temperature.

1. Click *Text*  .
2. Accept the default name and click **OK**.
3. Apply the following settings:

Tab	Setting	Value
Definition	Text String	Time Elapsed:
	Embed Auto Annotation	(Selected)[a]
	Type	Time Value

[a]The full text string should now be `Time Elapsed: <aa>`. The `<aa>` string represents the location where text is to be substituted.

4. Click **More** to add a second line of text to the text object.
5. Apply the following settings:

Tab	Setting	Value
Definition	Text String (the second one)	Sensor Temperature:
	Embed Auto Annotation	(Selected)
	Type	Expression
	Expression	TSensor
Appearance	Height	0.03

6. Click **Apply**.
7. Ensure that the visibility for `Text 1` is turned on.

The text label appears in the viewer, near the top.

19.7.2. Creating an Animation

1. Ensure that the view is set to **Isometric View (Z up)**.
2. Click *Timestep Selector*  .
3. Double-click the first time value (0 s).
4. In the toolbar at the top of the window click *Animation*  .
5. In the **Animation** dialog box, select the **Keyframe Animation** option.
6. Click *New*  to create `KeyframeNo1`.
7. Select `KeyframeNo1`, then set **# of Frames** to `200`, then press **Enter** while in the **# of Frames** box.

Tip

Be sure to press **Enter** and confirm that the new number appears in the list before continuing.

This will place 200 intermediate frames between the first and (yet to be created) second key frames, for a total of 202 frames. This will produce an animation lasting about 8.4 s since the frame rate will be 24 frames per second. Since there are 76 unique frames, each frame will be shown at least once.

8. Use the **Timestep Selector** to load the last time value (225 s).

9. In the **Animation** dialog box, click *New*  to create `KeyframeNo2`.

   The **# of Frames** parameter has no effect for the last keyframe, so leave it at the default value.

10. Click *More Animation Options*  to expand the **Animation** dialog box.
11. Select **Save Movie**.
12. Set **Format** to `MPEG1`.
13. Specify a file name for the movie file.
14. Click the **Options** button.
15. Change **Image Size** to `720 x 480` (or a similar resolution).
16. Click the **Advanced** tab, and note the **Quality** setting. If your movie player cannot play the resulting MPEG, you can try using the **Low** or **Custom** quality settings.
17. Click **OK**.
18. Click *To Beginning*  to rewind the active key frame to `KeyframeNo1`.
19. Click *Save animation state*  and save the animation to a file. This will enable you to quickly restore the animation in case you want to make changes. Animations are not restored by loading ordinary state files (those with the .cst extension).
20. Click *Play the animation*  .
21. If prompted to overwrite an existing movie, click **Overwrite**.

    The animation plays and builds an `mpg` file.

22. When you have finished, quit CFD-Post.

19.8. Further Steps

- This tutorial uses an aggressive flow rate of air, a coarse mesh, large time steps, and a low cap on the maximum number of coefficient loops per time step. Running this tutorial with a flow rate of air that is closer to 5 changes of air per hour (0.03 m^3 s^{-1}), a finer mesh, smaller time steps, and a larger cap on the maximum number of coefficient loops, will produce more accurate results.
- Running this simulation for a longer total time will allow you to see more on/off cycles of the air conditioner.

Chapter 20: Combustion and Radiation in a Can Combustor

This tutorial includes:

20.1. Tutorial Features

In this tutorial you will learn about:

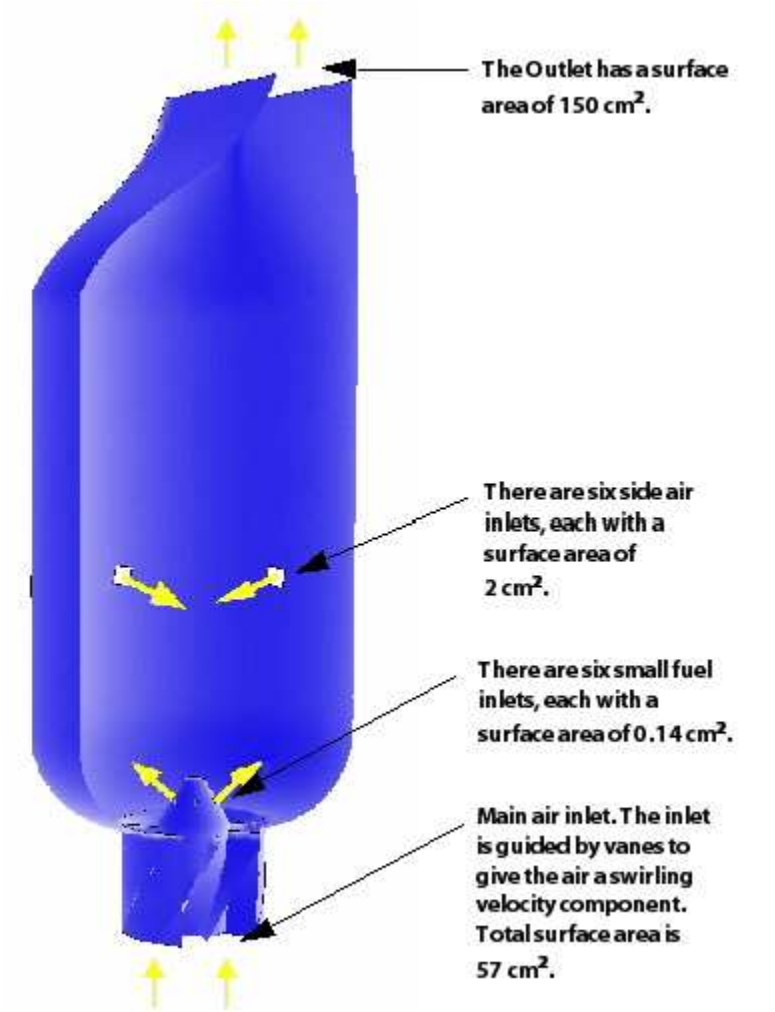
- Setting Up a Combustion Model in CFX-Pre.
- Using a Reacting Mixture.
- Using the Eddy Dissipation combustion model.
- Using the P1 radiation model.
- Creating thin surfaces for the inlet vanes.
- Using the Laminar Flamelet model.
- Generating a CFX-RIF library.
- Using the Discrete Transfer radiation model.
- Using chemistry post-processing.
- Changing object color maps in CFD-Post to prepare a grayscale image.
- Using the function calculator in CFD-Post.
- Creating a vector plot in CFD-Post.

Component	Feature	Details
CFX-Pre	User Mode	General Mode
	Analysis Type	Steady State
	Fluid Type	Reacting Mixture
	Domain Type	Single Domain
	Turbulence Model	k-Epsilon
	Heat Transfer	Thermal Energy

Component	Feature	Details
	Combustion	
	Radiation	
	Boundary Conditions	Inlet (Subsonic)
		Outlet (Subsonic)
		Wall: No-Slip
		Wall: Adiabatic
		Wall: Thin Surface
	Timestep	Physical Time Scale
CFD-Post	Plots	Outline Plot (Wireframe)
		Sampling Plane
		Slice Plane
		Vector
	Other	Changing the Color Range
		Color map
		Legend
		Quantitative Calculation

20.2. Overview of the Problem to Solve

The can combustor is a feature of the gas turbine engine. Arranged around a central annulus, can combustors are designed to minimize emissions, burn very efficiently and keep wall temperatures as low as possible. This tutorial is designed to give a qualitative impression of the flow and temperature distributions inside a can combustor that burns methane in air. The basic geometry is shown below with a section of the outer wall cut away.



This tutorial demonstrates two combustion and radiation model combinations. The simulation in the first part of this tutorial uses the Eddy Dissipation combustion model and the P1 radiation model. The simulation in the second part of this tutorial uses the Laminar Flamelet combustion model with a CFX-RIF-generated chemistry library, and the Discrete Transfer radiation model.

Different radiation models are used in this tutorial for demonstration purposes; the radiation models are independent of the combustion models.

See Which Model is the Most Appropriate? in the *CFX-Solver Modeling Guide* for a comparison of the available combustion models:

- Eddy Dissipation Model (EDM)
- Finite Rate Chemistry Model (FRC)
- Combined EDM/FRC
- Laminar Flamelet Model
- Burning Velocity

Due to the fact that the fuel (methane) and oxidizer (air) undergo "fast" combustion (whereby the combustion rate is dominated by the rate of mixing of the materials), the Finite Rate Chemistry model is not a suitable combustion model for the combustor in this tutorial. The Combined EDM/FRC model capability is a superset of the Eddy Dissipation model capability, and has no benefit over the Eddy Dissipation model in this case.

In fact, the convergence behavior of the Combined EDM/FRC model may be worse than that of the Eddy Dissipation model.

The Eddy Dissipation model, the Laminar Flamelet model, and the Burning Velocity model are suitable for modeling "fast" combustion. The Burning Velocity model capability is a superset of the Flamelet model capability, with the extra capability of being able to handle premixed fuel/oxidizer. Because the combustor in this tutorial does not use premixed fuel/oxidizer, the extra capability of the Burning Velocity model is not required and therefore it is sufficient to use the Flamelet model.

The Eddy Dissipation model tracks each individual chemical species (except for the constraint material) with its own transport equation. This model is flexible in that you can readily add new materials, such as additional fuels, to the simulation without complications. A limitation of this model is that radical or intermediate species, such as CO, cannot be calculated with adequate accuracy. This may lead to over-prediction of flame temperature, in particular in fuel-rich regions.

The Laminar Flamelet model can simulate the products of incomplete combustion; for this reason, it generally provides a more accurate solution than the Eddy Dissipation model. One drawback of the Flamelet model is that it requires the availability of a flamelet library suited for the required fuel/oxidizer combination over the pressure and temperature ranges of interest. In this tutorial, you will be generating a CFX-RIF library.

NO is modeled in a similar way in both parts of this tutorial. The only difference is in how the O-radical concentration is obtained for the 'Thermal NO' formation step:

- Eddy Dissipation model: The O-radical is not a component of the mixture; instead, its concentration is estimated using the O_2 concentration and temperature.
- Flamelet model: The O-radical concentration is calculated from the flamelet library, where its concentration information is directly available.

For details on the Thermal NO mechanism, see Thermal NO in the *CFX-Solver Theory Guide*.

The Flamelet configuration will utilize chemistry post-processing in solving for the concentration of NO. This post-processing will be one-way coupled to the main solution so that the formation of NO will be driven by the main solution without affecting the latter. This approach is appropriate for this simulation because the mass fraction and reaction rate of NO are sufficiently small, so the effect on the main solution is negligible.

20.3. Using Eddy Dissipation and P1 Models

In this first part of the tutorial, you will create a simulation that uses the Eddy Dissipation combustion model and the P1 radiation model. If you want to use the Flamelet combustion model and Discrete Transfer radiation model instead, see *Defining a Simulation Using Laminar Flamelet and Discrete Transfer Models in CFX-Pre* (p. 357), otherwise continue from this point.

20.4. Before You Begin

If this is the first tutorial you are working with, it is important to review the following topics before beginning:

- *Preparing a Working Directory and Starting ANSYS CFX in Standalone Mode* (p. 1)
- *Running ANSYS CFX Tutorials in ANSYS Workbench* (p. 2)
- *Changing the Display Colors* (p. 5)
- *Playing a Tutorial Session File* (p. 4)

20.5. Starting CFX-Pre

1. Prepare the working directory using the following files in the `examples` directory:
   - `CombustorMesh.gtm`
   - `CombustorEDM.pre`
   - `CombustorFlamelet.pre`
   - `CombustorEDM.cfx`
2. Set the working directory and start CFX-Pre.

   For details, see *Preparing a Working Directory and Starting ANSYS CFX in Standalone Mode* (p. 1)

20.6. Defining a Case Using Eddy Dissipation and P1 Models in CFX-Pre

If you want to set up the simulation automatically using a tutorial session file, run `CombustorEDM.pre`. For details, see *Playing a Tutorial Session File* (p. 4). Then proceed to *Obtaining a Solution Using CFX-Solver Manager* (p. 354).

If you want to set up the simulation manually, proceed to the following steps:

You will first define a domain that includes a variable composition mixture. These mixtures are used to model combusting and reacting flows in CFX.

1. In CFX-Pre, select **File** > **New Case**.
2. Select **General** and click **OK**.
3. Select **File** > **Save Case As**.
4. Under **File name**, type `CombustorEDM`.
5. Click **Save**.
6. If prompted, click **Overwrite**.

   This file is provided in the tutorial directory and may exist in your working directory if you have copied it there.

20.6.1. Importing the Mesh

1. Right-click `Mesh` and select **Import Mesh** > **CFX Mesh**. The **Import Mesh** dialog box appears.
2. Apply the following setting

Setting	Value
File name	CombustorMesh.gtm

3. Click **Open**.

20.6.2. Creating a Reacting Mixture

To allow combustion modeling, you must create a variable composition mixture.

20.6.2.1. To create the variable composition mixture

1. In the **Outline** tree, right-click **Materials** > **Insert** > **Material**.
2. Set the name to `Methane Air Mixture` and click **OK**.
3. Apply the following settings

Tab	Setting	Value
Basic Settings	Option	Reacting Mixture
	Material Group	Gas Phase Combustion
	Reactions List	Methane Air WD1 NO PDF [1 (p. 348)]
Mixture Properties	Mixture Properties	(Selected)
	Mixture Properties > Radiation Properties > Refractive Index	(Selected) [2 (p. 348)]
	Mixture Properties > Radiation Properties > Absorption Coefficient	(Selected)
	Mixture Properties > Radiation Properties > Scattering Coefficient	(Selected)

Footnotes

1. The `Methane Air WD1 NO PDF`reaction specifies complete combustion of the fuel into its products in a single-step reaction. The formation of NO is also modeled and occurs in an additional reaction step. Click ... to display the **Reactions List** dialog box, then click *Import Library Data* and select the appropriate reaction to import.
2. Setting the radiation properties explicitly will significantly shorten the solution time because the CFX-Solver will not have to calculate radiation mixture properties.

4. Click **OK**.

20.6.3. Creating the Domain

If `Default Domain` does not currently appear under **Flow Analysis 1** in the **Outline** tree:

Edit `Case Options` > `General` in the **Outline** tree view and ensure that **Automatic Default Domain** is turned on.

You now need to edit `Default Domain` so that it is representative of the Eddy Dissipation combustion and P 1 radiation models.

1. Double-click `Default Domain` and apply the following settings

Tab	Setting	Value
Basic Settings	Location and Type > Location	B152, B153, B154, B155, B156

Tab	Setting	Value
	Fluid and Particle Definitions	Fluid 1
	Fluid and Particle Definitions > Fluid 1 > Material	Methane Air Mixture
	Domain Models > Pressure > Reference Pressure	1 [atm] [1 (p. 349)]
Fluid Models	Heat Transfer > Option	Thermal Energy
	Combustion > Option	Eddy Dissipation
	Combustion > Eddy Dissipation Model Coefficient B	(Selected)
	Combustion > Eddy Dissipation Model Coefficient B > EDM Coeff. B	0.5 [2 (p. 349)]
	Thermal Radiation > Option	P 1
	Component Models > Component > N2	(Selected)
	Component Models > N2 > Option	Constraint

Footnotes

1. It is important to set a realistic reference pressure in this tutorial because the components of `Methane Air Mixture` are ideal gases.
2. This includes a simple model for partial premixing effects by turning on the Product Limiter. When it is selected, non-zero initial values are required for the products. The products limiter is not recommended for multi-step eddy dissipation reactions, and so is set for this single step reaction only.

2. Click **OK**.

20.6.4. Creating the Boundaries

20.6.4.1. Fuel Inlet Boundary

1. Create a new boundary by clicking *Boundary* and set the name to `fuelin`.
2. Apply the following settings:

Tab	Setting	Value
Basic Settings	Boundary Type	Inlet
	Location	fuelin
Boundary Details	Mass and Momentum > Normal Speed	40 [m s^-1]
	Heat Transfer > Static Temperature	300 [K]
	Component Details > CH4	(Selected)
	Component Details > CH4 > Mass Fraction	1.0

3. Click **OK**.

20.6.4.2. Bottom Air Inlet Boundary

Two separate boundary conditions will be applied for the incoming air. The first is at the base of the can combustor. The can combustor employs vanes downstream of the bottom air inlet to give the incoming air a swirling velocity.

1. Create a new boundary named `airin`.
2. Apply the following settings

Tab	Setting	Value
Basic Settings	Boundary Type	Inlet
	Location	airin
Boundary Details	Mass and Momentum > Normal Speed	10 [m s^-1]
	Heat Transfer > Static Temperature	300 [K]
	Component Details > O2	(Selected)
	Component Details > O2 > Mass Fraction	0.232 [1 (p. 350)]

Footnote

1. The remaining mass fraction at the inlet will be made up from the constraint component, N2.

3. Click **OK**.

20.6.4.3. Side Air Inlet Boundary

The secondary air inlets are located on the side of the vessel and introduce extra air to aid combustion.

1. Create a new boundary named `secairin`.
2. Apply the following settings

Tab	Setting	Value
Basic Settings	Boundary Type	Inlet
	Location	secairin
Boundary Details	Mass and Momentum > Normal Speed	6 [m s^-1]
	Heat Transfer > Static Temperature	300 [K]
	Component Details > O2	(Selected)
	Component Details > O2 > Mass Fraction	0.232 [1 (p. 350)]

Footnote

1. The remaining mass fraction at the inlet will be made up from the constraint component, N2.

3. Click **OK**.

20.6.4.4. Outlet Boundary

1. Create a new boundary named `out`.
2. Apply the following settings

Tab	Setting	Value
Basic Settings	Boundary Type	Outlet
	Location	out
Boundary Details	Mass and Momentum > Option	Average Static Pressure
	Mass and Momentum > Relative Pressure	0 [Pa]

3. Click **OK**.

20.6.4.5. Vanes Boundary

The vanes above the main air inlet are to be modeled as thin surfaces. To create a vane as a thin surface in CFX-Pre, you must specify a wall boundary on each side of the vanes.

You will first create a new region which contains one side of each of the eight vanes.

1. Create a new composite region by selecting **Insert** > **Regions** > **Composite Region**.
2. Set the name of this composite region to `Vane Surfaces`.
3. Apply the following settings

Tab	Setting	Value
Basic Settings	Dimension (Filter)	2D [1 (p. 351)]
	Region List	F129.152, F132.152, F136.152, F138.152, F141.152, F145.152, F147.152, F150.152 [2 (p. 351)]

Footnote

1. This will filter out the 3D regions, leaving only 2D regions
2. Click *Multi-select from extended list*  to open the Selection Dialog box, then hold the **Ctrl** key while selecting each item in this list. Click **OK**.

4. Click **OK**.
5. Create another composite region named `Vane Surfaces Other Side`.
6. Apply the following settings

Tab	Setting	Value
Basic Settings	Dimension (Filter)	2D
	Region List	F129.153, F132.153, F136.154, F138.154, F141.155, F145.155, F147.156, F150.156

7. Click **OK**.
8. Create a new boundary named `vanes`.
9. Apply the following settings

Tab	Setting	Value
Basic Settings	Boundary Type	Wall
	Location	`Vane Surfaces`, `Vane Surfaces Other Side` [1 (p. 352)]

Footnote

1. Click *Multi-select from extended list* ... to open the Selection Dialog box, then hold the **Ctrl** key while selecting both `Vane Surfaces` and `Vane Surfaces Other Side` from this list. Click **OK**.

10. Click **OK**.

20.6.4.6. Default Wall Boundary

The default boundary for any undefined surface in CFX-Pre is a no-slip, smooth, adiabatic wall.

- For radiation purposes, the wall is assumed to be a perfectly absorbing and emitting surface (emissivity = 1).
- The wall is non-catalytic, i.e., it does not take part in the reaction.

Since this tutorial serves as a basic model, heat transfer through the wall is neglected. As a result, no further boundary conditions need to be defined.

20.6.5. Setting Initial Values

1. Click *Global Initialization* .
2. Apply the following settings

Tab	Setting	Value
Global Settings	Initial Conditions > Cartesian Velocity Components > Option	Automatic with Value

Tab	Setting	Value
	Initial Conditions > Cartesian Velocity Components > U	0 [m s^-1]
	Initial Conditions > Cartesian Velocity Components > V	0 [m s^-1]
	Initial Conditions > Cartesian Velocity Components > W	5 [m s^-1]
	Initial Conditions > Component Details > O2	(Selected)
	Initial Conditions > Component Details > O2 > Option	Automatic with Value
	Initial Conditions > Component Details > O2 > Mass Fraction	0.232 [1 (p. 353)]
	Initial Conditions > Component Details > CO2	(Selected)
	Initial Conditions > Component Details > CO2 > Option	Automatic with Value
	Initial Conditions > Component Details > CO2 > Mass Fraction	0.01
	Initial Conditions > Component Details > H2O	(Selected)
	Initial Conditions > Component Details > H2O > Option	Automatic with Value
	Initial Conditions > Component Details > H2O > Mass Fraction	0.01

Footnote

1. The initial conditions assume the domain consists mainly of air and the fraction of oxygen in air is 0.232. A small mass fraction of reaction products (CO_2 and H_2O) is needed for the EDM model to initiate combustion.

3. Click **OK**.

20.6.6. Setting Solver Control

1. Click *Solver Control*  .
2. Apply the following settings

Tab	Setting	Value
Basic Settings	Convergence Control > Max. Iterations	100
	Convergence Control > Fluid Timescale Control > Timescale Control	Physical Timescale
	Convergence Control > Fluid Timescale Control > Physical Timescale	0.025 [s]

3. Click **OK**.

20.6.7. Writing the CFX-Solver Input (.def) File

1. Click *Define Run*  .
2. Apply the following settings:

Setting	Value
File name	CombustorEDM.def

3. Click **Save**.

   CFX-Solver Manager automatically starts and, on the **Define Run** dialog box, the **Solver Input File** is set.
4. If using Standalone Mode, quit CFX-Pre, saving the simulation (`.cfx`) file.

20.7. Obtaining a Solution Using CFX-Solver Manager

The CFX-Solver Manager will be launched after CFX-Pre saves the CFX-Solver input file. You will be able to obtain a solution to the CFD problem by following the instructions below.

Note

If a fine mesh is used for a formal quantitative analysis of the flow in the combustor, the solution time will be significantly longer than for the coarse mesh. You can run the simulation in parallel to reduce the solution time. For details, see *Obtaining a Solution in Parallel* (p. 118).

1. Ensure **Define Run** is displayed.

   CFX-Solver Input File should be set to `CombustorEDM.def`.
2. Click **Start Run**.

   CFX-Solver runs and attempts to obtain a solution. This can take a long time depending on your system. Eventually a dialog box is displayed stating that the run has finished.
3. Select the check box next to **Post-Process Results** when the completion message appears at the end of the run.
4. If using Standalone Mode, select the check box next to **Shut down CFX-Solver Manager**.
5. Click **OK**.

20.8. Viewing the Results in CFD-Post

When CFD-Post opens, experiment with the **Edge Angle** setting for the `Wireframe` object and the various rotation and zoom features in order to place the geometry in a sensible position. A setting of about `8.25` should result in a detailed enough geometry for this exercise.

20.8.1. Temperature Within the Domain

1. Right-click a blank area in the viewer and select **Predefined Camera** > **View From +Y**.
2. Create a new plane named `Plane 1`.

3. Apply the following settings

Tab	Setting	Value
Geometry	Definition > Method	ZX Plane
Color	Mode	Variable
	Mode > Variable	Temperature

4. Click **Apply**.

The large area of high temperature through most of the vessel is due to forced convection.

Note

Later in this tutorial (see *Defining a Simulation Using Laminar Flamelet and Discrete Transfer Models in CFX-Pre* (p. 357)), the Laminar Flamelet combustion model will be used to simulate the combustion again, resulting in an even higher concentration of high temperatures throughout the combustor.

20.8.2. The NO Concentration in the Combustor

In the next step you will color `Plane 1` by the mass fraction of NO to view the distribution of NO within the domain. The NO concentration is highest in the high temperature region close to the outlet of the domain.

1. Modify the plane named `Plane 1`.
2. Apply the following settings

Tab	Setting	Value
Color	Mode > Variable	NO.Mass Fraction

3. Click **Apply**.

20.8.3. Printing a Greyscale Graphic

Here you will change the color map (for `Plane 1`) to a greyscale map. The result will be a plot with different levels of grey representing different mass fractions of NO. This technique is especially useful for printing, to a black and white printer, any image that contains a color map. Conversion to greyscale by conventional means (i.e., using graphics software, or letting the printer do the conversion) will generally cause color legends to change to a non-linear distribution of levels of grey.

1. Modify the plane named `Plane 1`.
2. Apply the following settings

Tab	Setting	Value
Color	Color Map	Inverse Greyscale

3. Click **Apply**.

20.8.4. Calculating NO Mass Fraction at the Outlet

The emission of pollutants into the atmosphere is always a design consideration for combustion applications. In the next step, you will calculate the mass fraction of NO in the outlet stream.

1. Select **Tools** > **Function Calculator** or click the **Calculators** tab and select **Function Calculator**.
2. Apply the following settings

Tab	Setting	Value
Function Calculator	Function	massFlowAve
	Location	out
	Variable	NO.Mass Fraction

3. Click **Calculate**.

A small amount of NO is released from the outlet of the combustor. This amount is lower than can normally be expected, and is mainly due to the coarse mesh and the short residence times in the combustor.

20.8.5. Viewing Flow Field

To investigate the reasons behind the efficiency of the combustion process, you will next look at the velocity vectors to show the flow field. You may notice a small recirculation in the center of the combustor. Running the problem with a finer mesh would show this region to be a larger recirculation zone. The coarseness of the mesh in this tutorial means that this region of flow is not accurately resolved.

1. Select the **Outline** tab.
2. Under **User Locations and Plots**, clear **Plane 1**.

   `Plane 1` is no longer visible.

3. Create a new vector by clicking *Vector*  .
4. Accept the default name of `Vector 1`.
5. Apply the following settings

Tab	Setting	Value
Geometry	Definition > Locations	Plane 1
Symbol	Symbol Size	2

6. Click **Apply**.
7. Create a new plane named `Plane 2`.
8. Apply the following settings

Tab	Setting	Value
Geometry	Definition > Method	XY Plane
	Definition > Z	0.03 [m]
	Plane Bounds > Type	Rectangular
	Plane Bounds > X Size	0.5 [m]

Tab	Setting	Value
	Plane Bounds > Y Size	0.5 [m]
	Plane Type > Sample	(Selected)
	Plane Type > X Samples	30
	Plane Type > Y Samples	30
Render	Show Faces	(Cleared)

9. Click **Apply**.
10. Modify `Vector 1`.
11. Apply the following setting

Tab	Setting	Value
Geometry	Definition > Locations	Plane 2

12. Click **Apply**.

To view the swirling velocity field, right-click in the viewer and select **Predefined Camera** > **View From -Z**

You may also want to turn off the wireframe visibility. In the region near the fuel and air inlets, the swirl component of momentum (theta direction) results in increased mixing with the surrounding fluid and a higher residence time in this region. As a result, more fuel is burned.

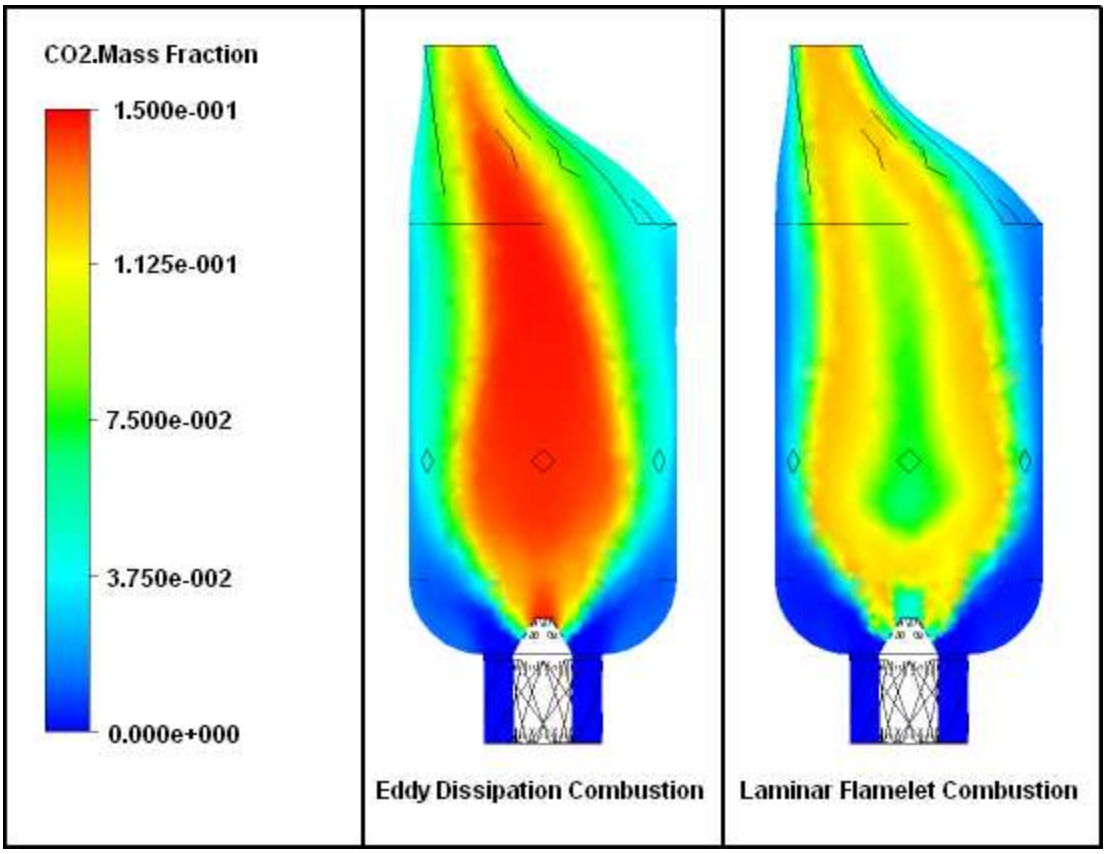
20.8.6. Viewing Radiation

Try examining the distribution of `Incident Radiation` and `Radiation Intensity` throughout the domain.

When you are finished, quit CFD-Post.

20.9. Defining a Simulation Using Laminar Flamelet and Discrete Transfer Models in CFX-Pre

In this second part of the tutorial, you will modify the simulation that was set up in the first part of the tutorial to use the Laminar Flamelet combustion model and the Discrete Transfer radiation model. Running the simulation a second time will demonstrate the differences in the combustion models, including the variance in carbon dioxide distribution, which is shown below.



20.9.1. Playing a Session File

If you want to set up the simulation automatically, run `CombustorFlamelet.pre` then continue to *Obtaining a Solution Using CFX-Solver Manager* (p. 364).

20.9.2. Creating a New Case

1. If you have not completed the first part of this tutorial, or otherwise do not have the simulation file from the first part, start CFX-Pre and then play the session file `CombustorEDM.pre`. The simulation file `CombustorEDM.cfx` will be created. Be sure to close the case once the session has been played.
2. If CFX-Pre is not already running, start it and load the simulation called `CombustorEDM.cfx`.

   The simulation from the first part of this tutorial is loaded.
3. Select **File** > **Save Case As**.
4. Save the simulation as `CombustorFlamelet.cfx`.

   This creates a separate simulation file which will be modified to use the Laminar Flamelet and Discrete Transfer models.

20.9.3. Removing Old Reactions

In the first part of this tutorial, the multi-step reaction responsible for the overall formation of NO, `NO Formation Methane PDF`, referred to the single-step reaction, `Thermal NO PDF`, for thermal NO formation. In this part of the tutorial, you will switch from using the `Thermal NO PDF` reaction to using the `Thermal NO O Radical PDF` reaction. The latter makes use of O radical information provided in

the flamelet library. The Thermal NO PDF reaction is less suitable because it approximates the O radical concentration.

Also in the first part of the tutorial, the material Methane Air Mixture referred to the multi-step reaction Methane Air WD1 NO PDF. In this part of the tutorial, this material will be changed to refer to a flamelet library, Methane300K, which you will generate using CFX-RIF.

Thus, in this section you are going to remove the Methane Air WD1 NO PDF and the Thermal NO PDF reactions because you will be replacing each of these for the flamelet configuration.

1. Expand the **Reactions** section in the **Outline** tree.
2. Right-click Methane Air WD1 NO PDF and select **Delete**.
3. Right-click Thermal NO PDF and select **Delete**.

Note

You will see a number of physics warnings appear at the bottom of the 3D Viewer. This is normal because you have just removed a reaction that the Methane Air Mixture depended upon. The warnings will be addressed when you generate a new CFX-RIF reaction and edit the Methane Air Mixture.

20.9.4. Importing a New Reaction

In this section, you will load the Thermal NO O Radical PDF reaction from a list of pre-generated reaction libraries in CFX-Pre. You will then adjust the NO Formation Methane PDF multi-step reaction to reference this reaction.

1. Right-click Reactions in the **Outline** tree and select **Import Library Data**.
2. Select Thermal NO O Radical PDF from the list of reaction libraries to import.
3. Click **OK** to import this reaction library into your simulation.
4. Right-click NO Formation Methane PDF in the **Outline** tree and select **Edit**.
5. Set **Reactions List** to Prompt NO Methane PDF, Thermal NO O Radical PDF.

Note

In order to select both reactions, you must click *Multi-select from extended list*  to open the Selection Dialog box, then hold the **Ctrl** key while selecting both Prompt NO Methane PDF and Thermal NO O Radical PDF from this list, and click **OK** to accept this selection.

6. Click **OK** to apply this change.

20.9.5. Generating the Flamelet Library

You will generate a flamelet library for the methane-air reaction using CFX-RIF. You will then use this library to modify the reacting mixture, Methane Air Mixture created in the first part of this tutorial.

1. In the **Outline** tree, right-click Reactions and select **Insert** > **CFX-RIF**.

2. Set the name to `CFX RIF CH4at300K` and click **OK**.
3. Apply the following settings:

Tab	Setting	Value
Basic Settings	Library File	Methane300K [1 (p. 360)]
	Fuel Module	CH4
	Kinetic Scheme	C1 mechanism without NOx [2 (p. 360)]
Boundary Conditions	Fuel Composition > Component Details > CH4 > Mass Fraction	1.0
	Fuel Composition > Component Details > N2 > Mass Fraction	0.0
	Oxidizer Composition > Component Details > N2 > Mass Fraction	0.767
	Oxidizer Composition > Component Details > O2 > Mass Fraction	0.233
	Pressure And Temperature > Fuel Temperature	300 [K]
	Pressure And Temperature > Oxidizer Temperature	300 [K]
	Pressure And Temperature > Reference Pressure	1 [atm] [3 (p. 360)]

Footnote

1. `Methane300K` will be used in the name of the flamelet library directory created inside of the working directory – for example, the folder might be named "Methane300K_001.dir".
2. Here you are choosing to generate the flamelet without the inclusion of NO. Instead, you will introduce chemistry post-processing for the modeling of NO when you modify the domain. This will allow the solver to run faster, as the corresponding transport equation will be solved using the final results from the main combustion reaction as a starting point, therefore requiring fewer iterations to converge.
3. The reference pressure here is *not* the same as the reference pressure set for the solver, but rather is the average expected operating pressure of the combustor.

4. Click **OK** to apply these settings.
5. In the **Outline** tree , right-click `CFX RIF CH4at300K` and select **Start CFX-RIF Generation**.

Note

An information message will appear shortly to tell you that the RIF process has successfully started, and initial data has been loaded into CFX-Pre. Click **OK** to continue.

Note

You will also see a warning in the bottom of the viewer informing you that the specified Flamelet Library file cannot be opened for the reaction `Methane300K`. Continue on with the tutorial. The library file (`Methane300K.fll`) will be generated as you work.

20.9.6. Modifying the Reacting Mixture

You will now use the CFX-RIF-generated library to modify the reacting mixture.

1. Expand the **Materials** section in the **Outline** tree.
2. Right-click `Methane Air Mixture` and select **Edit**.
3. Apply the following settings

Tab	Setting	Value
Basic Settings	Reactions List	`Methane 300K`

4. Click **OK** to apply this change.

20.9.7. Modifying the Default Domain

1. Double-click `Default Domain`.
2. Apply the following settings

Tab	Setting	Value
Fluid Models	Combustion > Option	PDF Flamelet
	Combustion > Chemistry Post Processing	(Selected)
	Combustion > Chemistry Post Processing > Materials List	NO
	Combustion > Chemistry Post Processing > Reactions List	NO Formation Methane PDF
	Thermal Radiation > Option	Discrete Transfer
	Component Models > Component > N2	(Selected)
	Component Models > Component > N2 > Option	Constraint
	Component Models > Component > NO	(Selected)
	Component Models > Component > NO > Option	Transport Equation
	Component Models > Component > (All Others)	(Selected one-at-a-time) [*1* **(p. 362)**]
	Component Models > Component > (All Others) > Option	Automatic

Footnote

1. Select these one at a time and verify each of them.

3. Click **OK** to apply these settings.

20.9.8. Modifying the Boundaries

20.9.8.1. Fuel Inlet Boundary

1. Right-click the boundary named `fuelin` and select **Edit**.
2. Apply the following settings

Tab	Setting	Value
Boundary Details	Mixture > Option	Fuel
	Component Details	NO
	Component Details > NO > Option	Mass Fraction
	Component Details > NO > Mass Fraction	0.0

3. Click **OK** to apply these settings.

20.9.8.2. Bottom Air Inlet Boundary

1. Right-click the boundary named `airin` and select **Edit**.
2. Apply the following settings

Tab	Setting	Value
Boundary Details	Mixture > Option	Oxidizer
	Component Details	NO
	Component Details > NO > Option	Mass Fraction
	Component Details > NO > Mass Fraction	0.0

3. Click **OK** to apply these settings.

20.9.8.3. Side Air Inlet Boundary

1. Right-click the boundary named `secairin` and select **Edit**.
2. Apply the following settings

Tab	Setting	Value
Boundary Details	Mixture > Option	Oxidizer
	Component Details	NO
	Component Details > NO > Option	Mass Fraction

Tab	Setting	Value
	Component Details > NO > Mass Fraction	0.0

3. Click **OK** to apply these settings.

Note

At this point, you have modified the domain and all necessary boundary conditions to match the changes made to `Methane Air Mixture`. Thus, the bottom window of the 3D Viewer should now be clear of all warnings and error messages.

20.9.9. Setting Initial Values

1. Click *Global Initialization*  .
2. Apply the following settings

Tab	Setting	Value
Global Settings	Initial Conditions > Component Details	NO
	Initial Conditions > Component Details > NO > Option	Automatic with Value
	Initial Conditions > Component Details > NO > Mass Fraction	0.0

3. Click **OK** to apply these settings.

20.9.10. Setting Solver Control

To reduce the amount of CPU time required for solving the radiation equations, you can select to solve them only every 10 iterations.

1. Click *Solver Control*  .
2. Apply the following settings

Tab	Setting	Value
Advanced Options	Dynamic Model Control > Global Dynamic Model Control	(Selected)
	Thermal Radiation Control	(Selected)
	Thermal Radiation Control > Iteration Interval	(Selected)
	Thermal Radiation Control > Iteration Interval > Iteration Interval	10

3. Click **OK** to apply these settings.

20.9.11. Writing the CFX-Solver Input (.def) File

1. Click *Define Run*  .
2. Apply the following settings:

Setting	Value
File name	CombustorFlamelet.def

3. Click **Save**.

   CFX-Solver Manager automatically starts and, on the **Define Run** dialog box, the **Solver Input File** is set.
4. If using Standalone Mode, quit CFX-Pre, saving the simulation (`.cfx`) file at your discretion.

20.10. Obtaining a Solution Using CFX-Solver Manager

When CFX-Solver Manager has started, you can obtain a solution to the CFD problem by using the following procedure:

1. Ensure that the flamelet library calculation has finished by checking your working directory for the presence of `Methane300K.fll`. The CFX-RIF generation process requires a couple of minutes from the time it is started.
2. Ensure **Define Run** is displayed.

   CFX-Solver Input File should be set to `CombustorFlamelet.def`.
3. Click **Start Run**.

   CFX-Solver runs and attempts to obtain a solution. This can take a long time depending on your system. Eventually a dialog box is displayed.
4. When CFX-Solver is finished, select the check box next to **Post-Process Results**.
5. If using Standalone Mode, select the check box next to **Shut down CFX-Solver Manager**.
6. Click **OK**.

20.11. Viewing the Results in CFD-Post

In this section, you will plot the Temperature in the Domain, the NO Concentration in the Combustor, and the CO Concentration. You will also use the **Function Calculator** to calculate the NO Concentration, and the CO Mass Fraction at the Outlet.

20.11.1. Viewing Temperature within the Domain

1. Create a new plane named `Plane 1`.
2. Apply the following settings

Tab	Setting	Value
Geometry	Definition > Method	ZX Plane
	Definition > Y	0

Tab	Setting	Value
Color	Mode	Variable
	Mode > Variable	Temperature

3. Click **Apply**.

20.11.2. Viewing the NO Concentration in the Combustor

1. Modify the plane named `Plane 1`.
2. Apply the following settings

Tab	Setting	Value
Color	Mode > Variable	NO.Mass Fraction

3. Click **Apply**.

20.11.3. Calculating NO Concentration

The next calculation shows the amount of NO at the outlet.

1. Select **Tools** > **Function Calculator** or click the **Calculators** tab and select **Function Calculator**.
2. Apply the following settings

Tab	Setting	Value
Function Calculator	Function	massFlowAve
	Location	out
	Variable	NO.Mass Fraction

3. Click **Calculate**.

20.11.4. Viewing CO Concentration

The next plot will show the concentration of CO (carbon monoxide), which is a by-product of incomplete combustion and is poisonous in significant concentrations. As you will see, the highest values are very close to the fuel inlet and in the regions of highest temperature.

1. Modify the plane named `Plane 1`.
2. Apply the following settings

Tab	Setting	Value
Color	Mode > Variable	CO.Mass Fraction
	Range	Local

3. Click **Apply**.

20.11.5. Calculating CO Mass Fraction at the Outlet

In the next step, you will calculate the mass fraction of CO in the outlet stream.

1. Select **Tools** > **Function Calculator** or click the **Calculators** tab and select **Function Calculator**.
2. Apply the following settings

Tab	Setting	Value
Function Calculator	Function	massFlowAve
	Location	out
	Variable	CO.Mass Fraction

3. Click **Calculate**.

   There is approximately 0.3% CO by mass in the outlet stream.

20.12. Further Postprocessing

1. Try putting some plots of your choice into the Viewer. You can plot the concentration of other species and compare values to those found for the Eddy Dissipation model.
2. Examine the distribution of `Incident Radiation` and `Radiation Intensity` throughout the domain.
3. Load one combustion model, then load the other using the **Keep current cases loaded** option in the **Load Results File** dialog box. You can compare both models in the viewer at once, in terms of mass fractions of various materials, as well as total temperature and other relevant measurements.

Chapter 21: Cavitation Around a Hydrofoil

This tutorial includes:

21.1. Tutorial Features

In this tutorial you will learn about:

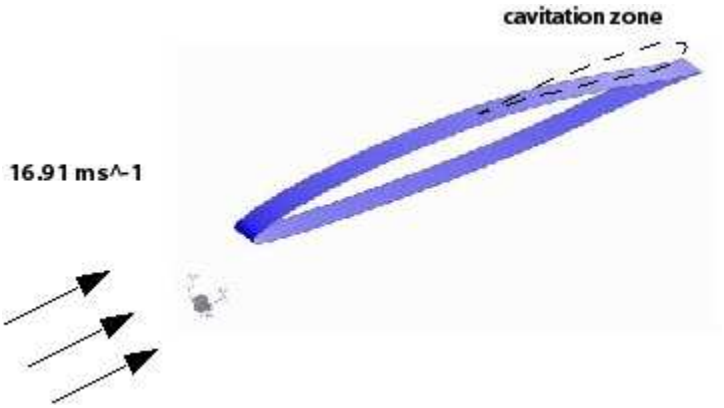
- Modeling flow with cavitation.
- Using vector reduction in CFD-Post to clarify a vector plot with many arrows.
- Importing and exporting data along a polyline.
- Plotting computed and experimental results.

Component	Feature	Details
CFX-Pre	User Mode	General Mode
	Analysis Type	Steady State
	Fluid Type	General Fluid
	Domain Type	Single Domain
	Turbulence Model	k-Epsilon
	Heat Transfer	Isothermal
	Multiphase	
	Boundary Conditions	Inlet (Subsonic)
		Outlet (Subsonic)
		Symmetry Plane
		Wall: No-Slip
		Wall: Free-Slip
	Timestep	Physical Time Scale
CFX-Solver Manager	Restart	
CFD-Post	Plots	Contour
		Line Locator

Component	Feature	Details
		Polyline
		Slice Plane
		Streamline
		Vector
	Other	Chart Creation
		Data Export
		Printing
		Title/Text
		Variable Details View

21.2. Overview of the Problem to Solve

This example demonstrates cavitation in the flow of water around a hydrofoil. A two-dimensional solution is obtained by modeling a thin slice of the hydrofoil and using two symmetry boundary conditions.



In this tutorial, an initial solution with no cavitation is generated to provide an accurate initial guess for a full cavitation solution, which is generated afterwards.

21.3. Before You Begin

If this is the first tutorial you are working with, it is important to review the following topics before beginning:

- *Preparing a Working Directory and Starting ANSYS CFX in Standalone Mode* (p. 1)
- *Running ANSYS CFX Tutorials in ANSYS Workbench* (p. 2)
- *Changing the Display Colors* (p. 5)
- *Playing a Tutorial Session File* (p. 4)

21.4. Starting CFX-Pre

1. Prepare the working directory using the following files in the `examples` directory:
   - `HydrofoilExperimentalCp.csv`
   - `HydrofoilGrid.def`
   - `HydrofoilIni.pre`

- `Hydrofoil.pre`
- `HydrofoilIni_001.res`

2. Set the working directory and start CFX-Pre.

   For details, see *Preparing a Working Directory and Starting ANSYS CFX in Standalone Mode* (p. 1)

21.5. Creating an Initial Simulation

This section describes the step-by-step definition of the flow physics in CFX-Pre.

21.5.1. Defining a Case in CFX-Pre

If you want to set up the simulation automatically using a tutorial session file, run `HydrofoilIni.pre`. For details, see *Playing a Tutorial Session File* (p. 4). Then proceed to *Obtaining an Initial Solution using CFX-Solver Manager* (p. 373).

If you want to set up the simulation manually, proceed to the following steps:

1. In CFX-Pre, select **File** > **New Case**.
2. Select **General** and click **OK**.
3. Select **File** > **Save Case As**.
4. Under **File name**, type `HydrofoilIni`.
5. Click **Save**.

21.5.2. Importing the Mesh

1. Right-click `Mesh` and select **Import Mesh** > **Other**. The **Import Mesh** dialog box appears.
2. Apply the following settings

Setting	Value
Files of type	CFX-Solver Input (*.def, *.res, *.trn, *.bak)
File name	HydrofoilGrid.def

3. Click **Open**.
4. Right-click a blank area in the viewer and select **Predefined Camera** > **View From -Z**.

21.5.3. Loading Materials

Since this tutorial uses `Water Vapour at 25 C` and `Water at 25 C`, you need to load these materials.

1. In the **Outline** tree view, right-click `Materials` and select **Import Library Data**.

   The **Select Library Data to Import** dialog box is displayed.
2. Expand **Water Data**.
3. Select both `Water Vapour at 25 C` and `Water at 25 C` by holding **Ctrl** when selecting.
4. Click **OK**.

21.5.4. Creating the Domain

The fluid domain used for this simulation contains liquid water and water vapor. The volume fractions are initially set so that the domain is filled entirely with liquid.

1. Edit `Case Options` > `General` in the **Outline** tree view and ensure that **Automatic Default Domain** is turned on. A domain named `Default Domain` should now appear under the `Simulation` > `Flow Analysis 1` branch.
2. Double-click `Default Domain`.
3. Under **Fluid and Particle Definitions**, delete `Fluid 1` and create a new fluid definition called `Liquid Water`.

   Use the  button to create another fluid named `Water Vapor`.

   Apply the following settings:

Tab	Setting	Value
Basic Settings	Fluid and Particle Definitions	Liquid Water
	Fluid and Particle Definitions > Liquid Water > Material	Water at 25 C
	Fluid and Particle Definitions	Water Vapor
	Fluid and Particle Definitions > Water Vapor > Material [1 (p. 370)]	Water Vapour at 25 C
	Domain Models > Pressure > Reference Pressure	0 [atm]
Fluid Models	Multiphase > Homogeneous Model	(Selected)
	Heat Transfer > Option	Isothermal
	Heat Transfer > Fluid Temperature	300 [K]
	Turbulence > Option	k-Epsilon

Footnote

1. These two fluids have consistent reference enthalpies.

4. Click **OK**.

21.5.5. Creating the Boundaries

The simulation requires inlet, outlet, wall and symmetry plane boundaries. The regions for these boundaries were imported with the grid file.

21.5.5.1. Inlet Boundary

1. Create a new boundary named `Inlet`.
2. Apply the following settings

Tab	Setting	Value
Basic Settings	Boundary Type	Inlet
	Location	IN
Boundary Details	Mass And Momentum > Normal Speed	16.91 [m s^-1]
	Turbulence > Option	Intensity and Length Scale
	Turbulence > Fractional Intensity	0.03
	Turbulence > Eddy Length Scale	0.0076 [m]
Fluid Values	Boundary Conditions	Liquid Water
	Boundary Conditions > Liquid Water > Volume Fraction > Volume Fraction	1
	Boundary Conditions	Water Vapor
	Boundary Conditions > Water Vapor > Volume Fraction > Volume Fraction	0

3. Click **OK**.

21.5.5.2. Outlet Boundary

1. Create a new boundary named `Outlet`.
2. Apply the following settings

Tab	Setting	Value
Basic Settings	Boundary Type	Outlet
	Location	OUT
Boundary Details	Mass And Momentum > Option	Static Pressure
	Mass And Momentum > Relative Pressure	51957 [Pa]

3. Click **OK**.

21.5.5.3. Free Slip Wall Boundary

1. Create a new boundary named `SlipWalls`.
2. Apply the following settings

Tab	Setting	Value
Basic Settings	Boundary Type	Wall
	Location	BOT, TOP
Boundary Details	Mass And Momentum > Option	Free Slip Wall

3. Click **OK**.

21.5.5.4. Symmetry Plane Boundaries

1. Create a new boundary named Sym1.
2. Apply the following settings

Tab	Setting	Value
Basic Settings	Boundary Type	Symmetry
	Location	SYM1

3. Click **OK**.

1. Create a new boundary named Sym2.
2. Apply the following settings

Tab	Setting	Value
Basic Settings	Boundary Type	Symmetry
	Location	SYM2

3. Click **OK**.

21.5.6. Setting Initial Values

1. Click *Global Initialization*  .
2. Apply the following settings

Tab	Setting	Value
Global Settings	Initial Conditions > Cartesian Velocity Components > Option	Automatic with Value
	Initial Conditions > Cartesian Velocity Components > U	16.91 [m s^-1]
	Initial Conditions > Cartesian Velocity Components > V	0 [m s^-1]
	Initial Conditions > Cartesian Velocity Components > W	0 [m s^-1]
Fluid Settings	Fluid Specific Initialization	Liquid Water
	Fluid Specific Initialization > Liquid Water > Initial Conditions > Volume Fraction > Option	Automatic with Value
	Fluid Specific Initialization > Liquid Water > Initial Conditions > Volume Fraction > Volume Fraction	1
	Fluid Specific Initialization	Water Vapor
	Fluid Specific Initialization > Water Vapor > Initial Conditions > Volume Fraction > Option	Automatic with Value
	Fluid Specific Initialization > Water Vapor > Initial Conditions > Volume Fraction > Volume Fraction	0

3. Click **OK**.

21.5.7. Setting Solver Control

1. Click *Solver Control*  .
2. Apply the following settings

Tab	Setting	Value
Basic Settings	Convergence Control > Max. Iterations	100
	Convergence Control > Fluid Timescale Control > Timescale Control	Physical Timescale
	Convergence Control > Fluid Timescale Control > Physical Timescale	0.01 [s]

Note

For the **Convergence Criteria**, an RMS value of at least 1e-05 is usually required for adequate convergence, but the default value is sufficient for demonstration purposes.

3. Click **OK**.

21.5.8. Writing the CFX-Solver Input (.def) File

1. Click *Define Run*  .
2. Apply the following settings

Setting	Value
File name	HydrofoilIni.def

3. Click **Save**.

   CFX-Solver Manager automatically starts and, on the **Define Run** dialog box, the **Solver Input File** is set.
4. Quit CFX-Pre, saving the simulation (`.cfx`) file at your discretion.

21.6. Obtaining an Initial Solution using CFX-Solver Manager

While the calculations proceed, you can see residual output for various equations in both the text area and the plot area. Use the tabs to switch between different plots (e.g., **Momentum and Mass**, **Turbulence Quantities**, etc.) in the plot area. You can view residual plots for the fluid and solid domains separately by editing the workspace properties.

1. Ensure that the **Define Run** dialog box is displayed.
2. Click **Start Run**.

   CFX-Solver runs and attempts to obtain a solution. This can take a long time depending on your system. Eventually a dialog box is displayed.

3. Select the check box next to **Post-Process Results** when the completion message appears at the end of the run.
4. If using Standalone Mode, select the check box next to **Shut down CFX-Solver Manager**.
5. Click **OK**.

21.7. Viewing the Results of the Initial Simulation

The following topics will be discussed:

21.7.1. Plotting Pressure Distribution Data

In this section, you will create a plot of the pressure coefficient distribution around the hydrofoil. The data will then be exported to a file for later comparison with data from the cavitating flow case, which will be run later in this tutorial.

1. Right-click a blank area in the viewer and select **Predefined Camera** > **View From -Z**.
2. Insert a new plane named `Slice`.
3. Apply the following settings

Tab	Setting	Value
Geometry	Definition > Method	XY Plane
	Definition > Z	5e-5 [m]
Render	Show Faces	(Cleared)

4. Click **Apply**.
5. Create a new polyline named `Foil` by selecting **Insert** > **Location** > **Polyline** from the main menu.
6. Apply the following settings

Tab	Setting	Value
Geometry	Method	Boundary Intersection
	Boundary List	Default Domain Default
	Intersect With	Slice

7. Click **Apply**.

   Zoom in on the center of the hydrofoil (near the cavity) to confirm the polyline wraps around the hydrofoil.
8. Define the following expressions, remembering to click **Apply** after entering each definition:

Name	Definition
PCoef	(Pressure-51957[Pa])/(0.5*996.2[kg m^-3]*16.91[m s^-1]^2)

Name	Definition
FoilChord	(X-minVal(X)@Foil)/(maxVal(X)@Foil-minVal(X)@Foil) [1 (p. 375)]

Footnote

1. This creates a normalized chord, measured in the X direction, ranging from 0 at the leading edge to 1 at the trailing edge of the hydrofoil.

9. Create a new variable named `Pressure Coefficient`.
10. Apply the following settings

Setting	Value
Method	Expression
Scalar	(Selected)
Expression	PCoef

11. Click **Apply**.
12. Create a new variable named `Chord`.
13. Apply the following settings

Setting	Value
Method	Expression
Scalar	(Selected)
Expression	FoilChord

14. Click **Apply**.

Note

Although the variables that were just created are only needed at points along the polyline, they exist throughout the domain.

Now that the variables `Chord` and `Pressure Coefficient` exist, they can be associated with the previously defined polyline (the locator) to form a chart line. This chart line will be added to the chart object, which is created next.

1. Select **Insert** > **Chart** from the main menu.
2. Set the name to `Pressure Coefficient Distribution`.
3. Apply the following settings

Tab	Setting	Value
General	Title	Pressure Coefficient Distribution

Tab	Setting	Value
Data Series	Name	Solver Cp
	Location	Foil
X Axis	Data Selection > Variable	Chord
	Axis Range > Determine ranges automatically	(Cleared)
	Axis Range > Min	0
	Axis Range > Max	1
	Axis Labels > Use data for axis labels	(Cleared)
	Axis Labels > Custom Label	Normalized Chord Position
Y Axis	Data Selection > Variable	Pressure Coefficient
	Axis Range > Determine Ranges Automatically	(Cleared)
	Axis Range > Min	-0.5
	Axis Range > Max	0.4
	Axis Range > Invert Axis	(Selected)
	Axis Labels > Use data for axis labels	(Cleared)
	Axis Labels > Custom Label	Pressure Coefficient

4. Click **Apply**.
5. The chart appears on the **Chart Viewer** tab.

21.7.2. Exporting Pressure Distribution Data

You will now export the chord and pressure coefficient data along the polyline. This data will be imported and used in a chart later in this tutorial for comparison with the results for when cavitation is present.

1. Select **File** > **Export** > **Export**. The **Export** dialog box appears
2. Apply the following settings

Tab	Setting	Value
Options	File	NoCavCpData.csv
	Locations	Foil
	Export Geometry Information	(Selected) [1 (p. 376)]
	Select Variables	Chord, Pressure Coefficient

Footnote

1. This causes X, Y, Z data to be included in the export file.

3. Click **Save**.

   The file `NoCavCpData.csv` will be written in the working directory.

21.7.3. Saving the Post-Processing State

If you are running CFD-Post in Standalone Mode, you will need to save the post-processing state for use later in this tutorial, as follows:

1. Select **File > Save State As**.
2. Under **File name** type `Cp_plot`, then click **Save**.

In the next part of this tutorial, the solver will be run with cavitation turned on. Similar post-processing follows, and the effect of cavitation on the pressure distribution around the hydrofoil will be illustrated in a chart.

21.8. Preparing a Simulation with Cavitation

Earlier in this tutorial, you ran a simulation without cavitation. The solution from that simulation will serve as the starting point for the next simulation, which involves cavitation.

21.8.1. Modifying the Initial Case in CFX-Pre

If you want to set up the simulation automatically and continue to *Obtaining a Cavitation Solution using CFX-Solver Manager* (p. 378), run `Hydrofoil.pre`.

1. If CFX-Pre is not already running, start it.
2. Select **File** > **Open Case**.
3. Select `HydrofoilIni_001.res` and click **Open**.
4. Save the case as `Hydrofoil.cfx`.

21.8.2. Adding Cavitation

1. Double-click `Default Domain` in the **Outline** tree view.
2. Apply the following settings

Tab	Setting	Value
Fluid Pair Models	Fluid Pairs > Liquid Water | Water Vapor > Mass Transfer > Option	Cavitation
	Fluid Pairs > Liquid Water | Water Vapor > Mass Transfer > Cavitation > Saturation Pressure	(Selected)
	Fluid Pairs > Liquid Water | Water Vapor > Mass Transfer > Cavitation > Saturation Pressure > Saturation Pressure	3574 [Pa] [1 (p. 377)]

Footnote

1. Although saturation pressure is optional, it must be set for this example. It is optional because saturation pressure can also be set by setting a homogeneous binary mixture, but one has not been used in this tutorial.

3. Click **OK**.

21.8.3. Modifying Solver Control

1. Click *Solver Control*  .
2. Apply the following settings

Tab	Setting	Value
Basic Settings	Convergence Control > Max. Iterations	150 [1 (p. 378)]

Footnote

1. This allows up to 150 further iterations, when run as a restart.

3. Click **OK**.

21.8.4. Modifying Execution Control

1. Click *Execution Control*  .
2. Apply the following settings

Tab	Setting	Value
Run Definition	Solver Input File	Hydrofoil.def [1 (p. 378)]

Footnote

1. You do not need to set the path unless you are planning on saving the solver file somewhere other than the working directory.

3. Confirm that the rest of the execution control settings are set appropriately.
4. Click **OK**.

21.8.5. Writing the CFX-Solver Input (.def) File

1. Click *Define Run*  .

   CFX-Solver Manager automatically starts and, on the **Define Run** dialog box, the **Solver Input File** and execution control settings are set.
2. If using Standalone Mode, quit CFX-Pre, saving the simulation (`.cfx`) file at your discretion.

21.9. Obtaining a Cavitation Solution using CFX-Solver Manager

1. Ensure the **Define Run** dialog box is displayed.

CFX-Solver Input File should be set to `Hydrofoil.def`.

2. Apply the following settings to set the **Initial Values File**.

Tab	Setting	Value
Run Definition	Initial Values Specification	Selected
	Initial Values Specification > Initial Values	Initial Values 1
	Initial Values Specification > Initial Values > Initial Values 1 Settings > File Name [1 (p. 379)]	`Hydrofoil-Ini_001.res`

Footnote

1. Click *Browse*  and select the file from the working directory.

This is the solution from the starting-point run.

3. Click **Start Run**.

CFX-Solver runs and attempts to obtain a solution. This can take a long time depending on your system. Eventually a dialog box is displayed.

4. Click **Yes** to post-process the results.
5. If using Standalone Mode, quit CFX-Solver Manager.

21.10. Viewing the Results of the Cavitation Simulation

You will restore the state file saved earlier in this tutorial while preventing the first solution (which has no cavitation) from loading. This will cause the plot of pressure distribution to use data from the currently loaded solution (which has cavitation). Data from the first solution will be added to the chart object by importing `NoCavCpData.csv` (the file that was exported earlier). A file containing experimental data will also be imported and added to the plot. The resulting chart will show all three sets of data (solver data with cavitation, solver data without cavitation, and experimental data).

Note

The experimental data is provided in `<CFXROOT>/examples/HydrofoilExperimentalCp.csv` which must be copied to your working directory before proceeding with this part of the tutorial.

Note

If using ANSYS Workbench, CFD-Post will already be in the state in which you left it in the first part of this tutorial. In this case, proceed to step 5 below.

1. Select **File** > **Load State**.
2. Clear **Load results**.
3. Select `Cp_plot.cst`.

4. Click **Open**.
5. Click the **Chart Viewer** tab.
6. Edit `Report` > `Pressure Coefficient Distribution`.
7. Click the **Data Series** tab.
8. Apply the following setting:

Tab	Setting	Value
Data Series	Name	Solver Cp - with cavitation

This reflects the fact that the user-defined variable `Pressure Coefficient` is now based on the current results.

9. Click **Apply**.

You will now add the chart line from the first simulation.

10. Create a new polyline named `NoCavCpPolyline`.
11. Apply the following setting:

Tab	Setting	Value
Geometry	File	NoCavCpData.csv

12. Click **Apply**.

The data in the file is used to create a polyline with values of `Pressure Coefficient` and `Chord` stored at each point on it.

13. Edit `Report` > `Pressure Coefficient Distribution`.
14. Click the **Data Series** tab.
15. Click **New**  .
16. Select `Series 2` from the list box.
17. Apply the following settings:

Tab	Setting	Value
Data Series	Name	Solver Cp - no cavitation
	Location	NoCavCpPolyline
	Custom Data Selection	(Selected)
	X Axis > Variable	Chord on NoCavCpPolyline
	Y Axis > Variable	Pressure Coefficient on NoCavCpPolyline

18. Click **Apply**.

The chart line (containing data from the first solution) is created, added to the chart object, and displayed on the **Chart Viewer** tab.

You will now add a chart line to show experimental results.

19. Click **New**  .
20. Apply the following settings

Tab	Setting	Value
Data Series	Name	Experimental Cp - with cavitation
	Data Source > File	(Selected)
	Data Source > File	HydrofoilExperiment-alCp.csv
Line Display	Line Display > Line Style	Automatic
	Line Display > Symbols	Rectangle

21. Click **Apply**.

    The chart line (containing experimental data) is created, added to the chart object, and displayed on the **Chart Viewer** tab.
22. If you want to save an image of the chart, select **File** > **Save Picture** from the main menu while the **Chart Viewer** tab is selected. This will allow you to save the chart to an image file.
23. When you are finished, close CFD-Post.

Chapter 22: Fluid Structure Interaction and Mesh Deformation

This tutorial includes:

22.1. Tutorial Features

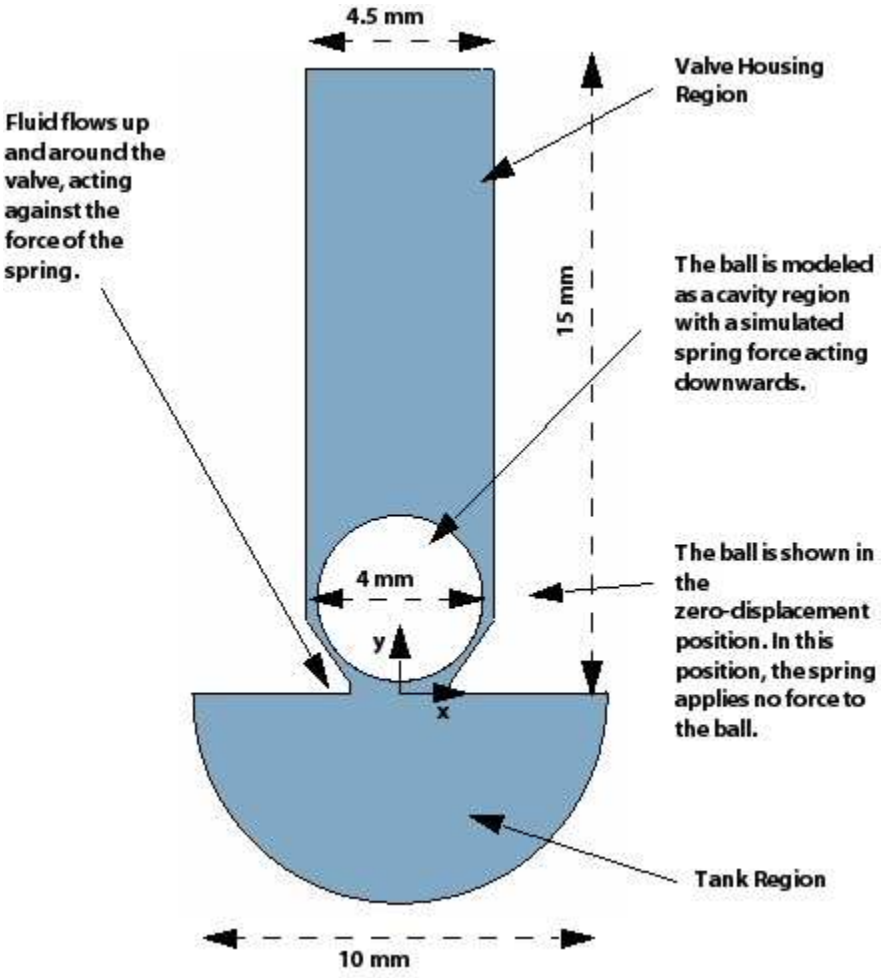
In this tutorial you will learn about:

- Mesh motion and deformation.
- Rigid body simulation.
- Fluid structure interaction (without modeling solid deformation).
- Animation creation.

Component	Feature	Details
CFX-Pre	User Mode	General Mode
	Analysis Type	Transient
	Fluid Type	General Fluid
	Domain Type	Single Domain
	Turbulence Model	k-Epsilon
	Heat Transfer	Isothermal
	Boundary Conditions	Opening
		Symmetry
		Wall
	Rigid Body	1 Degree of Freedom
	Mesh Motion	Unspecified
		Stationary
		Rigid Body Solution
CFD-Post	Plots	Slice Plane
		Point
		Vector Plot
		Animation

22.2. Overview of the Problem to Solve

This tutorial uses an example of a ball check valve to demonstrate two-way Fluid-Structure Interaction (FSI) between a ball and a fluid, as well as mesh deformation capabilities using ANSYS CFX. A sketch of the geometry, modeled in this tutorial as a 2-D slice (0.1 mm thick), is shown below.



Check valves are commonly used to enforce unidirectional flow of liquids and act as pressure-relieving devices. The check valve for this tutorial contains a ball connected to a spring with a stiffness constant of 300 N/m. The ball is made of steel with a density of 7800 kg/m^3 and is represented as a cavity region in the mesh with a diameter of 4 mm. Initially the center of mass of the ball is located at the coordinate point (0, 0.0023, 5e-05); this point is the spring origin, and all forces that interact with the ball are assumed to pass through this point. The tank region, located below the valve housing, is filled with Methanol (CH4O) at 25°C. High pressure from the liquid at the tank opening (6 atm relative pressure) causes the ball to move up, thus allowing the fluid to escape through the valve to the atmosphere at an absolute pressure of 1 atm. The forces on the ball are: the force due to the spring (not shown in the figure) and the force due to fluid flow. Gravity is neglected here for simplicity. The spring pushes the ball downward to oppose the force of the pressure when the ball is raised above its initial position. The pressure variation causes the ball to oscillate along the Y-axis as a result of a dynamic imbalance in the forces. The ball eventually stops oscillating when the forces acting on it are in equilibrium.

In this tutorial the deformation of the ball itself is not modeled; mesh deformation is employed to modify the mesh as the ball moves. A rigid body simulation is used to predict the motion of the ball, and will be

based on the forces that act on it. For further details on rigid body capabilities within ANSYS CFX, refer to Rigid Bodies in the *CFX-Pre User's Guide*.

22.3. Before You Begin

If this is the first tutorial you are working with, it is important to review the following topics before beginning:

- *Preparing a Working Directory and Starting ANSYS CFX in Standalone Mode* (p. 1)
- *Running ANSYS CFX Tutorials in ANSYS Workbench* (p. 2)
- *Changing the Display Colors* (p. 5)
- *Playing a Tutorial Session File* (p. 4)

22.4. Starting CFX-Pre

1. Copy the following files from the `<install_dir>\examples` directory into your working directory:
   - `ValveFSI.pre`
   - `ValveFSI.out`
2. Set the working directory in the CFX-Launcher and start CFX-Pre.

   For details, see *Preparing a Working Directory and Starting ANSYS CFX in Standalone Mode* (p. 1).

22.5. Defining a FSI Simulation in CFX-Pre

This section describes the step-by-step definition of the flow physics in CFX-Pre.

If you want to set up the simulation automatically using a tutorial session file, run `ValveFSI.pre`. For details, see *Playing a Tutorial Session File* (p. 4). Then proceed to *Obtaining a Solution Using CFX-Solver Manager* (p. 396).

If you want to set up the simulation manually, proceed to the following steps:

1. In CFX-Pre, select **File** > **New Case**.
2. Select **General** and click **OK**.
3. Select **File** > **Save Case As**.
4. Under **File name**, type `ValveFSI`.
5. Click **Save**.

22.5.1. Importing the Mesh

1. In the **Outline** tree view, right-click `Mesh` and select **Import Mesh** > **Other**. The **Import Mesh** dialog box appears.
2. Apply the following settings:

Setting	Value
Files of type	PATRAN Neutral
File name	ValveFSI.out

Setting	Value
Options > Mesh Units	mm [1 (p. 386)]

Footnote

1. This mesh was created using units of millimeters; however the units are not stored with this type of mesh. Set **Mesh Units** to `mm` when importing the mesh into CFX-Pre so that the mesh remains the intended size.

3. Click **Open**.

22.5.2. Defining a Transient Simulation

1. Click *Analysis Type*  .
2. Apply the following settings:

Tab	Setting	Value
Basic Settings	Analysis Type > Option	Transient
	Analysis Type > Time Duration > Option	Total Time
	Analysis Type > Time Duration > Total Time	7.5e-3 [s]
	Analysis Type > Time Steps > Option	Timesteps
	Analysis Type > Time Steps > Timesteps	5.0e-5 [s]
	Analysis Type > Initial Time > Option	Automatic with Value
	Analysis Type > Initial Time > Time	0 [s]

3. Click **OK**.

Note

You may ignore the physics validation message regarding the lack of definition of transient results files. You will set up the transient results files later.

22.5.3. Editing the Domain

In this section you will create the fluid domain, define the fluid and enable mesh motion.

1. If `Default Domain` does not currently appear under **Flow Analysis 1** in the **Outline** tree,

   edit `Case Options` > `General` in the **Outline** tree view and ensure that **Automatic Default Domain** is turned on and click **OK**.
2. In the tree view, right-click `Default Domain` and select **Edit**.

3. Apply the following settings:

Tab	Setting	Value
Basic Settings	Location and Type > Location	CV3D REGION, CV3D SUB [1 (p. 388)]
	Location and Type > Domain Type	Fluid Domain
	Fluid and Particle Definitions	Fluid 1
	Fluid and Particle Definitions > Fluid 1 > Material	Methanol CH4O [2 (p. 388)]
	Domain Models > Pressure > Reference Pressure	1 [atm]
	Domain Models > Mesh Deformation > Option	Regions of Motion Specified [3 (p. 388)]
	Domain Models > Mesh Deformation > Mesh Motion Model > Option	Displacement Diffusion [4 (p. 388)] [5 (p. 388)]
	Domain Models > Mesh Deformation > Mesh Motion Model > Mesh Stiffness > Option	Increase near Small Volumes
	Domain Models > Mesh Deformation > Mesh Motion Model > Mesh Stiffness > Model Exponent	10
Fluid Models	Heat Transfer > Option	Isothermal
	Heat Transfer > Fluid Temperature	25 [C]

Footnotes

1. Click the *Multi-select from extended list* icon ... to open the **Selection Dialog** dialog box, then hold the **Ctrl** key while selecting both `CV3D REGION` and `CV3D SUB` from this list. Click **OK**.
2. To make Methanol an available option:
   a. Click the *Select from extended list* icon ... to open the **Material** dialog box.
   b. Click the *Import Library Data* icon to open the **Select Library Data to Import** dialog box.
   c. In that dialog box, expand `Constant Property Liquids` in the tree, select `Methanol CH4O` and click **OK**.
   d. Select `Methanol CH4O` in the **Material** dialog box and click **OK**.
3. The `Regions of Motion Specified` option permits boundaries and subdomains to move, and makes mesh motion settings available.
4. To see the additional mesh motion settings, you may need to click *Roll Down* + located beside **Mesh Motion Model**.
5. The Displacement Diffusion model for mesh motion preserves the relative mesh distribution of the initial mesh.

4. Click **OK**.

22.5.4. Creating a Coordinate Frame

In this section, a secondary coordinate system will be created to define the center of mass of the ball. This secondary coordinate system will be used to define certain parameters of the rigid body in the next section.

1. In the **Outline** tree view, right-click **Coordinate Frames** and select **Insert** > **Coordinate Frame**.
2. Set the name to `Coord 1` and click **OK**.
3. Apply the following settings:

Tab	Setting	Value
Basic Settings	Option	Axis Points
	Origin	(0, 0.0023, 5e-05)
	Z Axis Point	(0, 0, 1)
	X-Z Plane Pt	(1, 0, 0)

4. Select **OK**.

22.5.5. Creating a Rigid Body

A rigid body is a non-deformable object described by physical parameters: mass, center of mass, moment of inertia, initial velocities and accelerations, and orientation. The rigid body solver utilizes the interacting

forces between the fluid and the rigid body and calculates the motion of the rigid body based upon the defined physical parameters. The rigid body may have up to six degrees of freedom (three translational and three rotational). You may also specify external forces and torques acting on the rigid body.

In this section, you will define a rigid body with 1 degree of freedom, translation in the Y-direction. The rigid body definition will be applied to the wall boundary of the ball to define its motion. Further, you will specify an external spring force by defining a spring constant and the initial origin of the spring; in this simulation the origin is the center of mass of the ball. The force caused by the tank pressure will cause an upward translation and the defined external spring force will resist this translation.

1. In the **Outline** tree view, right-click **Flow Analysis 1** and select **Insert** > **Rigid Body**.
2. Set the name to `rigidBall` and click **OK**.
3. Apply the following settings:

Tab	Setting	Value
Basic Settings	Mass	9.802e-6 [kg]
	Location	BALL
	Coord Frame	Coord 1
	Mass Moment of Inertia > XX Component	0 [kg m^2] [1 (p. 390)]
	Mass Moment of Inertia > YY Component	0 [kg m^2]
	Mass Moment of Inertia > ZZ Component	0 [kg m^2]
	Mass Moment of Inertia > XY Component	0 [kg m^2]
	Mass Moment of Inertia > XZ Component	0 [kg m^2]
	Mass Moment of Inertia > YZ Component	0 [kg m^2]
Dynamics	External Force Definitions	Create new external force named `Spring Force` [2 (p. 390)]
	External Force Definitions > Spring Force > Option	Spring
	External Force Definitions > Spring Force > Linear Spring Origin > X Component	0 [m]
	External Force Definitions > Spring Force > Linear Spring Origin > Y Component	0 [m]
	External Force Definitions > Spring Force > Linear Spring Origin > Z Component	0 [m]
	External Force Definitions > Spring Force > Linear Spring Constant > X Component	0 [N m^-1]

Tab	Setting	Value
	External Force Definitions > Spring Force > Linear Spring Constant > Y Component	300 [N m^-1]
	External Force Definitions > Spring Force > Linear Spring Constant > Z Component	0 [N m^-1]
	Degrees of Freedom	(Selected)
	Degrees of Freedom > Translational Degrees of Freedom	(Selected)
	Degrees of Freedom > Translational Degrees of Freedom > Option	Y axis
	Degrees of Freedom > Rotational Degrees of Freedom	(Selected)
	Degrees of Freedom > Rotational Degrees of Freedom > Option	None

Footnotes

1. The **Mass Moment of Inertia** settings can have any values; they have no effect on the simulation because the rigid body has only a singular, translational, degree of freedom.
2. To create a new item, you must first click the *Add new item* icon, then enter the name as required and click **OK**.

4. Click **OK**.

22.5.6. Creating the Subdomain

1. Select **Insert** > **Subdomain** from the main menu or click *Subdomain* .
2. Set the subdomain name to Tank and click **OK**.
3. Apply the following settings:

Tab	Setting	Value
Basic Settings	Location	CV3D SUB
Mesh Motion	Mesh Motion > Option	Stationary [1 (p. 390)]

Footnote

1. The stationary option for the tank volume (subdomain) ensures that the mesh does not fold at the sharp corners that exist where the valve joins the tank.

4. Click **OK**.

22.5.7. Creating the Boundaries

In the following subsections, you will create the required boundary conditions, specifying the appropriate mesh motion option for each.

In this tutorial, mesh motion specifications are applied to two and three dimensional regions of the domain. For example, the Ball boundary specifies the mesh motion in the form of the rigid body solution. However, mesh motion specifications are also used in this tutorial to help ensure that the mesh does not fold, as set for the Tank subdomain earlier in the tutorial, and the TankOpen boundary below. Two regions, VALVE HIGHX and VALVE LOWX, remain at the default boundary condition: smooth, no slip walls and no mesh motion (stationary).

22.5.7.1. Ball Boundary

1. Create a new boundary named Ball.
2. Apply the following settings:

Tab	Setting	Value
Basic Settings	Boundary Type	Wall
	Location	BALL
Boundary Details	Mesh Motion > Option	Rigid Body Solution
	Mesh Motion > Rigid Body	rigidBall
	Mass And Momentum > Option	No Slip Wall
	Mass And Momentum > Wall Velocity Relative To	(Selected)
	Mass And Momentum > Wall Velocity Relative To > Wall Vel. Rel. To	Mesh Motion

3. Click **OK**.

22.5.7.2. Symmetry Boundary

Because a 2D representation of the flow field is being modeled (using a 3D mesh, one element thick in the Z-direction), you must create symmetry boundaries on the low and high Z 2D regions of the mesh.

1. Create a new boundary named Sym.
2. Apply the following settings:

Tab	Setting	Value
Basic Settings	Boundary Type	Symmetry
	Location	SYMP1, SYMP2 [1 (p. 391)]
Boundary Details	Mesh Motion > Option	Unspecified

Footnotes

1. Hold the **Ctrl** key while selecting both SYMP1 and SYMP2 from the list.

3. Click **OK**.

22.5.7.3. Vertical Valve Wall Boundary

1. Create a new boundary named `ValveVertWalls`.
2. Apply the following settings:

Tab	Setting	Value
Basic Settings	Boundary Type	Wall
	Location	VPIPE HIGHX, VPIPE LOWX [1 (p. 392)]
Boundary Details	Mesh Motion > Option	Unspecified [2 (p. 392)]
	Mass And Momentum > Option	No Slip Wall
	Mass And Momentum > Wall Velocity Relative To	(Selected)
	Mass And Momentum > Wall Velocity Relative To > Wall Vel. Rel. To	Boundary Frame

Footnote

1. Hold the **Ctrl** key while selecting both `VPIPE HIGHX` and `VPIPE LOWX` from the list.
2. The `Unspecified` setting allows the mesh nodes to move freely. The motion of the mesh points on this boundary will be strongly influenced by the motion of the ball. Because the ball moves vertically, the surrounding mesh nodes should also move vertically, at a similar rate to the ball. This mesh motion specification helps to preserve the quality of the mesh on the upper surface of the ball.

3. Click **OK**.

22.5.7.4. Tank Opening Boundary

1. Create a new boundary named `TankOpen`.
2. Apply the following settings:

Tab	Setting	Value
Basic Settings	Boundary Type	Opening
	Location	BOTTOM
Boundary Details	Mesh Motion > Option	Stationary [1 (p. 393)]
	Mass And Momentum > Option	Entrainment
	Mass And Momentum > Relative Pressure	6 [atm] [2 (p. 393)]
	Turbulence > Option	Zero Gradient

Footnotes

1. The stationary option for the tank opening prevents the mesh nodes on this boundary from moving. If the tank opening had unspecified mesh motion, these mesh nodes would move vertically and separate from the non-vertical parts of the boundary.
2. As defined in the problem description. Note the units for this setting.

3. Click **OK**.

22.5.7.5. Valve Opening Boundary

1. Create a new boundary named `ValveOpen`.
2. Apply the following settings:

Tab	Setting	Value
Basic Settings	Boundary Type	Opening
	Location	TOP
Boundary Details	Mesh Motion > Option	Stationary [1 (p. 393)]
	Mass And Momentum > Option	Entrainment
	Mass And Momentum > Relative Pressure	0 [atm] [2 (p. 393)]
	Turbulence > Option	Zero Gradient

Footnotes

1. The stationary option for the valve opening prevents the mesh nodes from moving.
2. This pressure value is relative to the fluid domain's reference pressure of 1 [atm].

3. Click **OK**.

Note

Opening boundary types are used to allow the flow to leave and reenter the domain. This behavior is expected due to the oscillatory motion of the ball and due to the potentially large region of flow recirculation that may occur downstream from the ball.

22.5.8. Setting Initial Values

Because a transient simulation is being modeled, initial values are required for all variables.

1. Click *Global Initialization*  .
2. Apply the following settings:

Tab	Setting	Value
Global Settings	Initial Conditions > Cartesian Velocity Components > U	0 [m s^-1]
	Initial Conditions > Cartesian Velocity Components > V	0.1 [m s^-1] [1 (p. 394)]
	Initial Conditions > Cartesian Velocity Components > W	0 [m s^-1]
	Initial Conditions > Static Pressure > Relative Pressure	0 [Pa]
	Initial Conditions >Turbulence > Option	Medium (Intensity = 5%)

Footnotes

1. This is an initial velocity to start a unidirectional fluid flow in the positive Y-direction and to prevent initial backflow in the check-valve, improving solution convergence. Better values of velocity could be derived from the steady state analysis (not considered for this tutorial).

3. Click **OK**.

22.5.9. Setting Solver Control

In this section you will edit the solver control settings to promote a quicker solution time and to enable the frequency of when the rigid body solver is executed.

1. Click *Solver Control*  .
2. Apply the following settings:

Tab	Setting	Value
Basic Settings	Transient Scheme > Option	Second Order Backward Euler
	Convergence Control > Max. Coeff. Loops	5
Rigid Body Control	Rigid Body Control	(Selected)
	Rigid Body Control > Rigid Body Solver Coupling Control > Update Frequency	Every Coefficient Loop [1 (p. 394)]

Footnotes

1. By setting the **Update Frequency** to `Every Coefficient Loop` you are telling CFX-Solver to call the rigid body solver during every coefficient loop within each time step.

3. Click **OK**.

22.5.10. Setting Output Control

This step sets up transient results files to be written at set intervals.

1. Click *Output Control*  .
2. Click the **Trn Results** tab.
3. In the **Transient Results** tree view, click *Add new item*  , set **Name** to `Transient Results 1`, and click **OK**.
4. Apply the following settings to `Transient Results 1`:

Setting	Value
Option	Selected Variables
Output Variables List	Pressure, Velocity [1 (p. 395)]
Output Variable Operators	(Selected)
Output Variable Operators > Output Var. Operators	All [2 (p. 395)]
Output Frequency > Option	Time Interval
Output Frequency > Time Interval	5.0e-5 [s]

Footnotes

1. Click *Multi-select from extended list*  beside the entry box, and make multiple selections in the `Output Variables List` by holding down the **Ctrl** key and clicking on the required variables.
2. This causes the gradients of the selected variables to be written to the transient results files.

5. Click the **Monitor** tab.
6. Select **Monitor Objects**.
7. Under **Monitor Points and Expressions**:
   1. Click *Add new item*  .
   2. Set **Name** to `Ball Displacement` and click **OK**.
   3. Set **Option** to `Expression`.
   4. Set **Expression Value** to `rbstate(Position Y)@rigidBall`.
8. Click **OK**.

22.5.11. Writing the CFX-Solver Input (.def) File

1. Click *Define Run*  .
2. Apply the following settings:

Setting	Value
File name	ValveFSI.def

3. Click **Save**.
4. CFX-Solver Manager starts automatically and, on the **Define Run** dialog box, the **Solver Input File** is set.
5. Quit CFX-Pre, saving the simulation (`.cfx`) file.

22.6. Obtaining a Solution Using CFX-Solver Manager

When CFX-Pre has shut down and the CFX-Solver Manager has started, obtain a solution to the CFD problem by following the instructions below.

1. Ensure **Define Run** is displayed.

   Solver Input File should be set to `ValveFSI.def`.
2. Click **Start Run**.

   CFX-Solver runs and attempts to obtain a solution. This can take a long time depending on your system.
3. While CFX-Solver Manager is running, you can check the progress of the monitor point you created in CFX-Pre by clicking the `User Points` tab in CFX-Solver Manager. The graph shows the Y position of the center of mass of the ball (in the global coordinate frame). Notice that the ball has a sinusoidal motion that diminishes in amplitude over time and that the maximum displacement of the ball occurs at around time step 17.
4. Select **Monitors** > **Rigid Body** > **Rigid Body Position** from the main menu. The position of the rigid body will be shown in the X, Y and Z directions relative to the global coordinate frame.

   Note

   This graph is identical to the graph obtained from under the `User Points` tab (although the scale may be different). Normally, creating the monitor point for position is redundant since the rigid body positions are calculated automatically — the monitor point was created in this tutorial to demonstrate the `rbstate` function.
5. Eventually a dialog box is displayed. Select the check box next to **Post-Process Results** when the completion message appears at the end of the run.
6. Click **OK**.

22.7. Viewing the Results in CFD-Post

In the following subsections, you will create a user location, point and vector plots, and an animation in CFD-Post. You will create an XY plane that lies midway between the two symmetry planes. The plane will

be used to show the mesh motion; it will also serve as the location for a vector plot that will be used in the animation.

22.7.1. Creating a Slice Plane

1. Right-click a blank area in the viewer and select **Predefined Camera** > **View From +Z**.
2. Select **Insert** > **Location** > **Plane** from the main menu. Accept the default name and click **OK**.
3. Apply the following settings:

Tab	Setting	Value
Geometry	Definition > Method	XY Plane
	Definition > Z	5e-05 [m]
Render	Show Faces	(Cleared)
	Show Mesh Lines	(Selected)

4. Click **Apply**.

22.7.2. Creating Points and a Vector Plot

1. Select **Insert** > **Location** > **Point** from the main menu. Accept the default name and click **OK**.
2. Apply the following settings:

Tab	Setting	Value
Geometry	Definition > Method	XYZ
	Point	(0, 0.0003, 0)
Symbol	Symbol	Crosshair
	Symbol Size	5

3. Click **Apply** to create the point.

   This is a reference point for the minimum Y value of the ball at time step 0. However the final time step is currently selected. This will be corrected in the proceeding steps.
4. Select **Insert** > **Location** > **Point** from the main menu. Accept the default name and click **OK**.
5. Apply the following settings:

Tab	Setting	Value
Geometry	Definition > Method	XYZ
	Point	(0, 0.001252, 0)
Symbol	Symbol	Crosshair
	Symbol Size	5

6. Click **Apply** to create the point.

This is a reference point for the minimum Y value of the ball in the positive Y-direction at the time of maximum displacement.

7. Click *Timestep Selector* and load the results for a few different time steps, selecting one entry at a time.

   For example, double-click rows with the step values of 0, 10, 20, 50, and 90 to see the ball in different positions. The mesh deformation will also be visible.

8. Create a new vector named `Vector 1`.
9. Apply the following settings:

Tab	Setting	Value
Geometry	Locations	Plane 1
	Variable	Velocity

10. Click **Apply** to show the vector plot in the 3D Viewer.

22.7.3. Creating an Animation

You will create an animation showing the velocity in the domain as the ball moves.

1. Turn off the visibility of `Plane 1` to better see the vector plot.
2. Click the *Timestep Selector* and load the 1st time step.
3. Click *Animate timesteps* in the *Timestep Selector* dialog box.
4. In the **Animation** dialog box, select the **Keyframe Animation** option.
5. Click *New* to create `KeyframeNo1`.
6. Select `KeyframeNo1`, then set **# of Frames** to `149`, then press **Enter** while the cursor is in the **# of Frames** box.

   Tip

   Be sure to press **Enter** and confirm that the new number appears in the list before continuing.

7. Use the **Timestep Selector** to load the last time step.
8. In the **Animation** dialog box, click *New* to create `KeyframeNo2`.

   Tip

   The **# of Frames** parameter has no effect on the last keyframe, so leave it at the default value.

9. Ensure that *More Animation Options* is pushed down to show more animation settings.
10. Select **Loop**.

11. Ensure that *Repeat forever*  (next to the **Repeat** setting) is not selected (not pushed down).
12. Click the **Options** button to open the **Animation Options** dialog box.
13. Apply the following settings:

Tab	Setting	Value
Options	Print Options > Image Size	720 x 480 (NTSC)
Advanced	MPEG Options > Quality	Custom
	MPEG Options > Variable Bit Rate	(Cleared)
	MPEG Options > Bit Rate	3000000 [1 (p. 399)]

Footnote

1. This limits the bit rate so that the movie will be playable in most players. You can lower this value if your player cannot process at this bit rate.

14. Click **OK**.
15. Select **Save Movie**.
16. Set **Format** to `MPEG1`.
17. Click *Browse*  (next to **Save Movie**).
18. Set **File name** to `ValveFSI.mpg`.

    If required, set the path to a different directory.
19. Click **Save**.

    The movie file name (including the path) has been set, but the animation has not yet been produced.
20. Click *To Beginning*  .

    This ensures that the animation will begin at the first keyframe.
21. After the first keyframe has been loaded, click *Play the animation*  .

    - The MPEG will be created as the animation proceeds.
    - This will be slow, since results for each time step will be loaded and objects will be created.
    - To view the movie file, you need to use a viewer that supports the MPEG format.

Note

To explore additional animation options, click the **Options** button. On the **Advanced** tab of the **Animation Options** dialog box, there is a **Save Frames As Image Files** check box. By selecting this check box, the JPEG or PPM files used to encode each frame of the movie will persist after movie creation; otherwise, they will be deleted.

22. Close the **Animation** dialog box when the animation is complete.

23. When you have finished, close the **Timestep Selector** dialog box and quit CFD-Post.

Chapter 23: Oscillating Plate with Two-Way Fluid-Structure Interaction

This tutorial includes:

23.1. Tutorial Features

In this tutorial you will learn about:

- Moving mesh
- Fluid-structure interaction (including modeling structural deformation using ANSYS)
- Running an ANSYS Multi-field (MFX) simulation
- Post-processing two results files simultaneously.

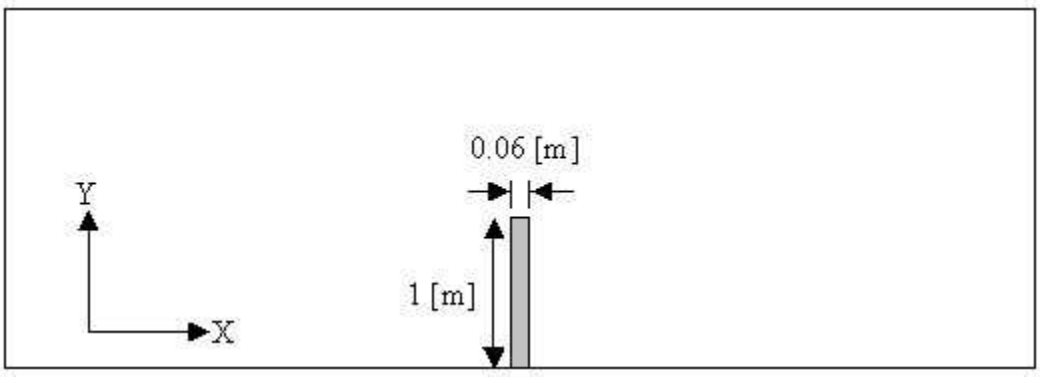
Component	Feature	Details
CFX-Pre	User Mode	General Mode
	Analysis Type	Transient
		ANSYS Multi-field
	Fluid Type	General Fluid
	Domain Type	Single Domain
	Turbulence Model	Laminar
	Heat Transfer	None
	Output Control	Monitor Points
		Transient Results File
	Boundary Conditions	Wall: Mesh Motion = ANSYS MultiField
		Wall: No Slip

Component	Feature	Details
		Wall: Adiabatic
	Timestep	Transient
CFD-Post	Plots	Animation
		Contour
		Vector

23.2. Overview of the Problem to Solve

This tutorial uses an example of an oscillating plate to demonstrate how to set up and run a simulation involving two-way Fluid-Structure Interaction (FSI) in ANSYS Workbench. In this tutorial, the structural physics is set up in the Transient Structural analysis system and the fluid physics is set up in Fluid Flow (CFX) analysis system, but both structural and fluid physics are solved together under the Solution cell of the Fluid system. Coupling between two analysis systems is required throughout the solution to model the interaction between structural and fluid systems as time progresses. The framework for the coupling is provided by the ANSYS Multi-field solver using the MFX setup.

The geometry consists of a 2D closed cavity and a thin plate, 1 m high, that is anchored to the bottom of the cavity as shown below:



An initial pressure of 100 Pa is applied to one side of the thin plate for 0.5 seconds in order to distort it. Once this pressure is released, the plate oscillates backwards and forwards as it attempts to regain its equilibrium (vertical) position. The surrounding fluid damps the plate oscillations, thereby decreasing the amplitude of oscillations with time. The CFX solver calculates how the fluid responds to the motion of the plate, and the ANSYS solver calculates how the plate deforms as a result of both the initial applied pressure and the pressure resulting from the presence of the fluid. Coupling between the two solvers is required since the structural deformation affects the fluid solution, and the fluid solution affects the structural deformation.

23.3. Before You Begin

- **Preparing a Working Directory**

  This tutorial uses the geometry file, `OscillatingPlate.agdb`, for setting up the project. This file is located in `<CFXROOT>/examples`, where `<CFXROOT>` is the installation directory for ANSYS CFX.

  Copy the supplied geometry file, `OscillatingPlate.agdb`, to a directory of your choice. This directory will be referred to as the *working directory* in this tutorial.

By working with a copy of the geometry file in a new directory, you prevent accidental changes to the file that came with your installation.

- **Changing the Appearance of ANSYS CFX Applications**

  If this is the first tutorial you are working with, see *Changing the Display Colors* (p. 5) for information on how to change the appearance of ANSYS CFX applications.

23.4. Creating the Project

1. Start ANSYS Workbench.

   To launch ANSYS Workbench on Windows, click the **Start** menu, then select **All Programs > ANSYS 13.0 > Workbench**. To launch ANSYS Workbench on Linux, open a command line interface, type the path to "runwb2" (for example, "~/ansys_inc/v130/Framework/bin/Linux64/runwb2"), then press **Enter**.

   The Project Schematic appears with an Unsaved Project. By default, ANSYS Workbench is configured to show the **Getting Started** dialog box that describes basic operations in ANSYS Workbench. Click the [X] icon to close this dialog box. To turn *on* or *off* this dialog box, select **Tools > Options** from the main menu and set **Project Management > Startup > Show Getting Started Dialog** as desired.

2. Select **File** > **Save** or click *Save*  .

   A **Save As** dialog box appears.

3. Select the path to your working directory to store files created during this tutorial.

   For details, see *Preparing a Working Directory* (p. 402).

4. Under **File name**, type `OscillatingPlate` and click **Save**.

   The project files and their associated folder locations appear under the **Files View**. To make the **Files View** visible, select **View > Files** from the main menu of ANSYS Workbench.

23.5. Adding Analysis Systems to the Project

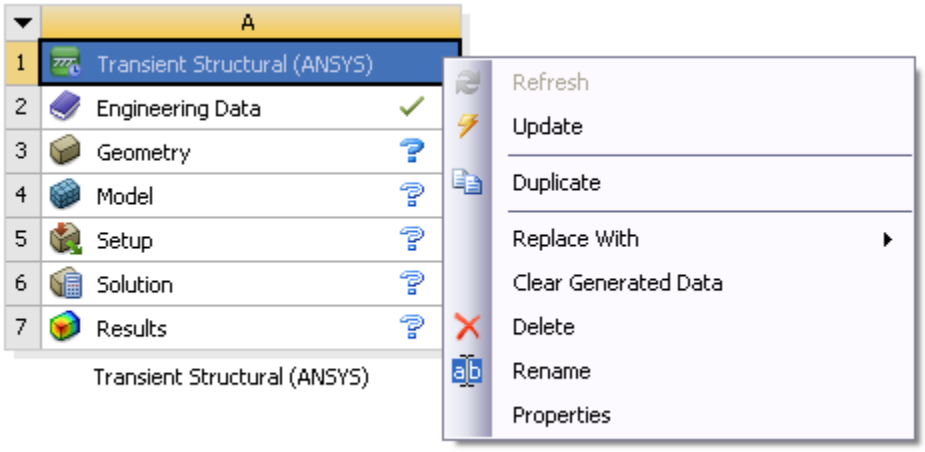
In ANSYS Workbench, a two-way FSI analysis can be performed by setting up a pair of coupled analysis systems, the pair consisting of a Transient Structural system and a Fluid Flow (CFX) system, as outlined in this section.

1. Expand the **Analysis Systems** option in the toolbox, located on the left side of the ANSYS Workbench window, and select the **Transient Structural** template. Double-click the template, or drag it onto the Project Schematic to create a standalone system.

   A Transient Structural system is added to the Project Schematic, with its name selected and ready to be renamed.

2. Type in the new name, `Structural`, to replace the selected text. This name will be used while referring to the Transient Structural system in this tutorial.

   If you missed seeing the selected text, right-click the first cell in the system and select **Rename** as shown in the following figure. The name will then be selected and ready to change.



3. Now right-click the **Setup** cell in the Structural system and select **Transfer Data to New > Fluid Flow (CFX)**.

   A Fluid Flow system, coupled to the ANSYS system, is added to the Project Schematic.

4. Change the name of this system to `Fluid`; this name will be used while referring to the Fluid Flow (CFX) system in this tutorial.

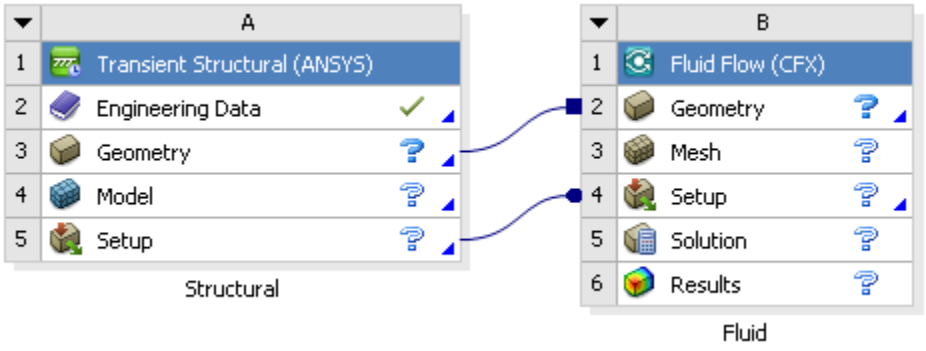
For this tutorial, the Solution and Results cells of the Structural system will be removed because they are not used for this two-way FSI analysis. This tutorial relies on the solution and results generated in the Fluid system, which you have already connected to the Structural system.

Remove the Solution and Results cells from the Structural system as follows:

1. In the Structural system, right-click the **Solution** cell and select **Delete**.
2. Click **OK** on the dialog box to confirm the deletion of the cell with the solution data from the Structural system.

   The Solution and Results cells disappear from the Structural system. The updated project is shown in *Figure 23.1* (p. 404).

Figure 23.1 Project setup for two-way FSI analysis



3. Now from the main menu, select **File > Save** to save the project setup.

The Structural and Fluid systems contain various cells. ANSYS Workbench provides visual indications of a cell's state at any given time via icons on the right side of each cell. In *Figure 23.1* (p. 404), most cells appear

with a blue question mark (?), indicating that cells need to be set up before continuing the analysis. As these cells are set up, the data transfer occurs from top to bottom. See *Understanding States* in ANSYS Workbench help for a description of various cell states.

Now the project is ready for further processing. A project with inter-connected systems enables you to perform the analysis by adding a new material, sharing the geometry, setting up the physics in the Structural system, and setting up the physics in the Fluid system. Later, the analysis will be performed in the Fluid system for solving and viewing results.

In *Figure 23.1* (p. 404), the **Engineering Data** cell appears in an *up-to-date* state, because a default material definition is already available for the project. However, the default material is not used in this tutorial. Thus, the next step in the analysis is to add a new material with properties desired for exhibiting an oscillation under the influence of external pressure, as outlined in *Overview of the Problem to Solve* (p. 402). The new material can be created using the Engineering Data application in ANSYS Workbench, as described in the next section.

23.6. Adding a New Material for the Project

This section describes how to create a new material named Plate, define its properties suitable for oscillation, and set it as the default material for the analysis.

1. On the Project Schematic, double-click the **Engineering Data** cell in the Structural system.

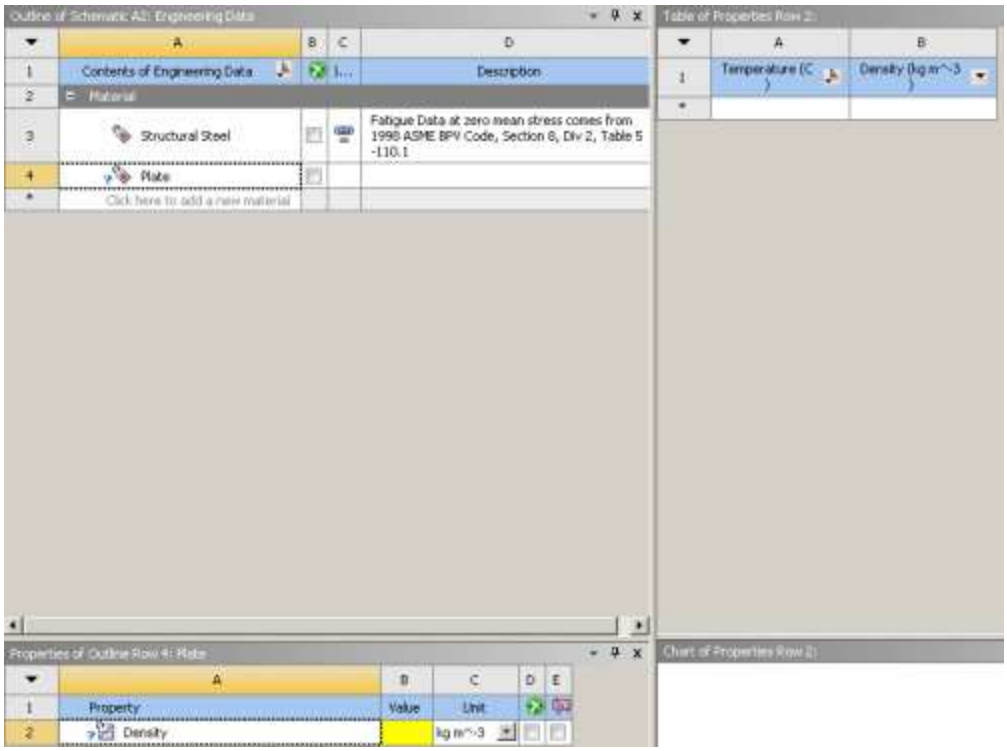
   The Outline and Properties windows appear.

2. In the **Outline of Schematic A2: Engineering Data** window, click the empty row at the bottom of the table to add a new material for the project. Type in the name `Plate`.

   Plate is created and appears with a blue question mark (?), indicating that plate properties need to be defined.

3. Now from the toolbox located on the left side of the ANSYS Workbench window, expand **Physical Properties**. Select **Density** and drag it onto the cell containing **Plate** in the **Outline of Schematic A2: Engineering Data** window.

   Density is added as the plate property in the **Properties of Outline Row 4: Plate** window, as shown in the following figure.



4. In the **Properties of Outline Row 4: Plate** window, set **Density** to `2550` [kg m^-3].
5. Similarly, from the **Linear Elastic** toolbox, drag **Isotropic Elasticity** onto **Plate** in the **Outline of Schematic A2: Engineering Data** window.

   Isotropic Elasticity is added as the plate property in the **Properties of Outline Row 4: Plate** window.
6. In the **Properties of Outline Row 4: Plate** window, expand **Isotropic Elasticity** by clicking on the plus sign. Now set **Young's Modulus** to `2.5e06` [Pa] and **Poisson's Ratio** to `0.35`.

Now the desired plate data is created and will be available to remaining cells in the Structural system. The next step is to set Plate as the default material for the analysis as outlined below:

1. In the **Outline of Schematic A2: Engineering Data** window, under **Material**, right-click **Plate** to open the shortcut menu.
2. In the shortcut menu, select **Default Solid Material For Model**.
3. Now from the main menu, select **File > Save** to save material settings to the project.

Now from the ANSYS Workbench toolbar, click **Return to Project** to close the Engineering Data workspace and return to the Project Schematic. The Outline and Properties windows disappear.

23.7. Adding Geometry to the Project

This section describes how to add geometry by importing an existing DesignModeler file and unsuppressing geometry parts in order to make the latter available for subsequent cells in the Structural and Fluid systems.

1. On the Project Schematic, right-click the **Geometry** cell in the Structural system and select **Import Geometry > Browse**.
2. In the **Open** dialog box, select `OscillatingPlate.agdb` from your working directory, and click **Open**.

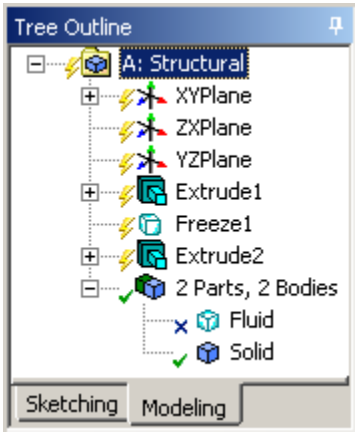
For details, see *Preparing a Working Directory* (p. 402).

3. In the Structural system, double-click the **Geometry** cell to edit the geometry using DesignModeler.

Note

Because the Geometry cell in the Structural system shares its content with the Geometry cell in the Fluid system, the latter cannot be edited.

In DesignModeler, the **Tree Outline** contains two bodies, Fluid and Solid, under the branch named **2 Parts, 2 Bodies**, as shown in the following figure.



The Fluid body appears in a suppressed state, shown with an x mark, implying that the body is not visible. When a body is suppressed in DesignModeler, its model data is not exported to subsequent cells in the analysis systems.

For this tutorial, all bodies will be unsuppressed in DesignModeler so that all geometry data is transferred to the subsequent cells in the Structural and Fluid systems. Later in the tutorial, the Fluid and Solid bodies will be suppressed selectively in the Structural and Fluid systems, respectively, before generating an appropriate structural or fluid mesh.

1. In the **Tree Outline**, right-click the **Fluid** body and select **Unsuppress Body**.

   The Fluid body is unsuppressed and a green check mark appears next to it in the **Tree Outline**.

2. In the **Tree Outline**, select the branch named **2 Parts, 2 Bodies**. Both the Fluid and Solid bodies should be visible in the Graphics window. Click *Zoom to Fit*  to view the entire model in the Graphics window.

This finishes the geometry setup for the project. Save these changes by selecting **File > Save Project** from the main menu in DesignModeler, and then select **File > Close DesignModeler** to return to the Project Schematic.

Now the updated geometry is available for both the Structural and Fluid systems.

23.8. Defining the Physics in the Mechanical Application

This section describes the step-by-step definition of the structural physics in the following sections:

23.8.1. Generating the Mesh for the Structural System

This section describes how to generate mesh for the Structural system.

1. On the Project Schematic, double-click the **Model** cell in the Structural system.

   The Mechanical application appears.
2. In the Mechanical application, expand **Project > Model > Geometry** in the tree view.

   Two geometries, Fluid and Solid, appear in the tree view. Click *Zoom to Fit*  to view the entire model in the Graphics window.

   For the Structural system, the mesh needs to be generated from the Solid body. As such, the Fluid body will be suppressed before the mesh generation operation.
3. Right-click the **Fluid** geometry and select **Suppress Body** from the shortcut menu.

   The Fluid body becomes suppressed and its status changes to an x mark. Click *Zoom to Fit*  to resize the model suitable for viewing in the Graphics window.
4. In the tree view, right-click **Mesh** and select **Generate Mesh** from the shortcut menu.

   The hex mesh is generated.

23.8.2. Assigning the Material to Geometry

1. In the Mechanical application, expand **Project > Model > Geometry** in the tree view and select **Solid**.

   The details of Solid appear in the **Details** view below the Outline tree view.
2. In the **Details** view, ensure **Material > Assignment** is set to `Plate`. Otherwise, click on the material name and use the arrow that appears next to the material name to make appropriate changes.

23.8.3. Basic Analysis Settings

This section outlines the steps to set up an ANSYS Multi-field run using the transient mechanical analysis, with a timestep of 0.1 s and a time duration of 5 s. For the given material properties of the plate, the time duration is chosen to allow the plate to oscillate just a few times, and the timestep is chosen to resolve those oscillations to a reasonable degree.

1. In the Mechanical application, expand **Project > Model > Transient** in the tree view and select **Analysis Settings**.

   The details of Analysis Settings appear in the **Details** view below the Outline tree view.
2. In the **Details** view, specify the following settings under **Step Controls**:

Note

Do not type in units while entering data for the time settings, **Time Step** and **Step End Time**.

- Set **Auto Time Stepping** to `Off`
- Set **Time Step** to `0.1`
- Set **Step End Time** to `5`.

23.8.4. Inserting Loads

The loads applied for the finite element analysis are equivalent to the boundary conditions in fluid analysis. In this section, you will set a fixed support, a fluid-solid interface, and a pressure load. On the surfaces of the plate that lie coincident with the symmetry planes, no loads are set. As a result, the default of an unconstrained condition will be applied on these surfaces. For this particular application, this is a reasonable approximation of the frictionless support that would otherwise be applied.

23.8.4.1. Fixed Support

The fixed support is required to hold the bottom of the thin plate in place.

1. In the Mechanical application, expand **Project > Model** and right-click **Transient** in the tree view and select **Insert > Fixed Support** from the shortcut menu.
2. Rotate the geometry using the *Rotate*  button so that the bottom (low-y) face of the solid is visible, then select *Face*  and click the *low-y* face.

   That face should be highlighted to indicate the selection.
3. In the **Details** view, click **Apply** to set the fixed support.

   The text next to the **Geometry** setting changes to `1 Face`.

   If the **Apply** button is not visible, select **Fixed Support** in the tree view and, in the **Details** view, click on the text next to the **Geometry** setting to make the **Apply** button re-appear.

23.8.4.2. Fluid-Solid Interface

The fluid-solid interface defines the interface between the fluid in the Fluid system and the solid in the Structural system. This interface is defined on regions in the structural model. Data is exchanged across this interface during the execution of the simulation.

1. In the Mechanical application, expand **Project > Model** and right-click **Transient** in the tree view and select **Insert > Fluid Solid Interface** from the shortcut menu.
2. Using the same face-selection procedure described earlier in *Fixed Support* (p. 409), select the three faces of the geometry that form the interface between the structural model and the fluid model (low-x, high-y and high-x faces) by holding down **Ctrl** to select multiple faces.

   Note that this load (fluid-solid interface) is automatically given an interface number of 1.

23.8.4.3. Pressure Load

The pressure load provides the initial additional pressure of `100 [Pa]` for the first 0.5 seconds of the simulation. It is defined using a step function.

1. In the Mechanical application, expand **Project > Model** and right-click **Transient** in the tree view and select **Insert > Pressure** from the shortcut menu.
2. Select the low-x face for **Geometry** and click **Apply**.
3. In the **Details** view, select **Magnitude**, and using the arrow that appears, select **Tabular** data.
4. Under **Tabular Data** at the bottom right of the Mechanical application window, set a pressure of `100` in the table row corresponding to a time of `0`.

   Note

   Do not type in units while entering the tabular data. The units for time and pressure in this table are the global units of `[s]` and `[Pa]`, respectively.

5. You now need to add two new rows to the table. This can be done by typing the new time and pressure data into the empty row at the bottom of the table, and rows will be automatically re-ordered based on the time value. Enter a pressure of `100` for a time value of `0.499`, and a pressure of `0` for a time value of `0.5`.

Tabular Data

	Steps	Time	Pressure
1	1	0.	100.
2	1	0.499	100.
3	1	0.5	0.
4	1	5.	0.
*			

This gives a step function for pressure that can be seen in the chart to the left of the table.

The settings for structural physics are now complete. Save these settings by selecting **File > Save Project** from the main menu, and select **File > Close Mechanical** to close the Mechanical application and return to the Project Schematic.

23.9. Completing the Setup for the Structural System

On the Project Schematic, the Setup cell in the Structural system appears in an *update-required* state. This section describes how to update the Setup cell in the Structural system.

1. In the Structural system, right-click the **Setup** cell and select **Update** from the shortcut menu.

   The status of the Setup cell changes to *up-to-date*. Now all cells in the Structural system should appear in an *up-to-date* state.
2. Now from the main menu, select **File > Save** to save the project.

This completes the setup for the Structural system. In the next section, the Fluid system will be set up.

As the Geometry cell is already up to date for both the Solid and Fluid systems, the next section begins with the setup of Mesh cell. Before generating mesh for the Fluid system, geometry faces will be grouped by creating Named Selections in the Meshing application as discussed in the next section.

23.10. Creating Named Selections on the Fluid Body

This section describes how to group geometry faces using Named Selections in the Meshing application. Later, when a mesh is generated from the model containing Named Selections, the grouped geometry faces are retained in the mesh and are accessible from within ANSYS CFX in the form of *Regions*.

1. On the Project Schematic, right-click the **Mesh** cell in the Fluid system and select **Edit** to open the model in the Meshing application.
2. In the Meshing application, expand **Project > Model > Geometry** in the tree view.

   Two items, Fluid and Solid, appear under the Geometry tree object.
3. Right-click the **Solid** body and select **Suppress Body** from the shortcut menu.

   The Solid body becomes suppressed and its status changes to an x mark.
4. In the graphics window, rotate the geometry using the *Rotate*  button so that the *high-z* face of the geometry is visible, then select *Face*  and click the *high-z* face.
5. Right-click in the viewer and select **Create Named Selection**.
6. Type in `Sym1` for the name of the selection group and click **OK**.
7. Following the same procedure, create `Sym2` by selecting the *low-z* face.
8. Finally, create a Named Selection named `Interface`, selecting the three faces that make contact with the solid geometry (the plate).

Note

Hold the **Ctrl** key to select multiple faces.

This finishes the creation of Named Selections on the Fluid body. Do not close the Meshing application yet; the tutorial continues to set up mesh settings and generate a mesh for the Fluid system in the next section.

23.11. Generating the Mesh for the Fluid System

1. In the Meshing application, expand **Project > Model > Geometry** in the tree view.

   Two geometries, Fluid and Solid, appear under Geometry.
2. In the tree view, expand **Project > Model > Mesh** and ensure that the **Mesh** branch does not contain any objects. Otherwise, right-click such objects and select **Delete** from the shortcut menu.
3. Ensure **Mesh** is selected in the tree view.

   The details of **Mesh** appear in the details view below the tree view.
4. In the details view, set **Sizing > Relevance Center** to `Medium`.

   This controls the grid resolution of the mesh.

5. Now in the tree view, right-click **Mesh** and select **Insert > Method** from the shortcut menu.

   Automatic Method is added to the **Mesh** branch in the tree view. In the details view, **Apply** and **Cancel** buttons appear next to the Geometry property.

6. Click anywhere on the geometry in the viewer to select the Fluid body.
7. In the details view, click **Apply** (for the **Geometry** property).

   Notice in the details view that **Scope > Geometry** is now set to `1 Body`.

8. In the details view, set the following mesh settings in the following order:

   1. Set **Definition > Method** to `Sweep`.
   2. Set **Free Face Mesh Type** to `All Tri`.
   3. Set **Sweep Num Divs** to `1`.

9. Now in the tree view, right-click **Mesh** and select **Update** from the shortcut menu.

   The mesh is generated.

10. This finishes the mesh generation for the Fluid system. From the main menu, select **File** > **Save Project** to save these changes to the project, and then select **File** > **Close Meshing** to return to the Project Schematic.

23.12. Defining the Physics and ANSYS Multi-field Settings in ANSYS CFX-Pre

This section describes the step-by-step definition of the flow physics and ANSYS Multi-field settings in the following sections:

23.12.1. Setting the Analysis Type

A transient ANSYS Multi-field run executes as a series of timesteps. In ANSYS CFX-Pre, the **Analysis Type** tab is used to enable both an ANSYS Multi-field run and to specify time-related settings for the coupled solver run. ANSYS CFX-Pre reads the ANSYS input file, which is automatically passed by ANSYS Workbench, in order to determine fluid-solid interfaces created in the Mechanical application.

Note

When ANSYS CFX-Pre is started, two errors will be displayed; these can be ignored because they will be fixed in the next steps of the tutorial.

1. On the Project Schematic, double-click the **Setup** cell in the Fluid system to launch the ANSYS CFX-Pre application.

2. In ANSYS CFX-Pre, click *Analysis Type* .

3. Apply the following settings:

Tab	Setting	Value
Basic Settings	External Solver Coupling > Option	ANSYS MultiField
	Coupling Time Control > Coupling Time Duration > Option	Total Time
	Coupling Time Control > Coupling Time Duration > Total Time	5 [s]
	Coupling Time Control > Coupling Time Steps > Option	Timesteps
	Coupling Time Control > Coupling Time Steps > Timesteps	0.1 [s]
	Analysis Type > Option	Transient
	Analysis Type > Time Duration > Option	Coupling Time Duration [a]
	Analysis Type > Time Steps > Option	Coupling Timesteps [a]
	Analysis Type > Initial Time > Option	Coupling Initial Time [a]

[a] Once the timesteps and time duration are specified for the ANSYS Multi-field run (coupling run), CFX automatically picks up these settings and it is not possible to set the timestep and time duration independently. Hence the only option available for **Time Duration** is `Coupling Time Duration`, and similarly for the related settings `Time Step` and `Initial Time`.

4. Click **OK**.

23.12.2. Creating the Fluid

A custom fluid is created with user-specified properties.

1. Click *Material* and set the name of the material to `Fluid`.

2. Apply the following settings:

Tab	Setting	Value
Basic Settings	Option	Pure Substance
	Thermodynamic State	(Selected)
	Thermodynamic State > Thermodynamic State	Liquid
Material Properties	Equation of State > Molar Mass	1 [kg kmol^-1] [a]
	Equation of State > Density	1 [kg m^-3] [b]
	Transport Properties > Dynamic Viscosity	(Selected)
	Transport Properties > Dynamic Viscosity > Dynamic Viscosity	0.2 [Pa s] [b]

[a]The molar mass is not used for this tutorial setup and has been set only for the completeness of the fluid property.

[b]The fluid properties are chosen to ensure that the plate generates a reasonable amplitude of vibration that doesn't decay too fast under the influence of fluid.

3. Click **OK**.

23.12.3. Creating the Domain

In order to allow ANSYS Solver to communicate mesh displacements to CFX-Solver, mesh motion must be activated in CFX.

1. Edit `Case Options` > `General` in the **Outline** tree view and ensure that **Automatic Default Domain** is turned on. A domain named `Default Domain` should now appear under the `Simulation` branch.
2. Double-click `Default Domain` and apply the following settings

Tab	Setting	Value
Basic Settings	Fluid and Particle Definitions	Fluid 1
	Fluid and Particle Definitions > Fluid 1 > Material	Fluid
	Domain Models > Pressure > Reference Pressure	1 [atm] [a]
	Domain Models > Mesh Deformation > Option	Regions of Motion Specified
Fluid Models	Heat Transfer > Option	None
	Turbulence > Option	None (Laminar)

[a]The reference pressure has no effect on this simulation so leave it as the default.

3. Click **OK**.

23.12.4. Creating the Boundaries

In addition to the symmetry conditions, this tutorial requires boundary conditions for an external boundary resulting from the fluid-solid interface as outlined below:

- *Fluid Solid External Boundary* (p. 414)
- *Symmetry Boundaries* (p. 415)

23.12.4.1. Fluid Solid External Boundary

The interface between ANSYS and CFX is considered as an external boundary in CFX-Solver with its mesh displacement being defined by the ANSYS Multi-field coupling process. This section outlines the steps to create a **Boundary Type** for CFX and specify a matching **ANSYS interface**. This specification sets up CFX-Solver to pass forces to ANSYS solver on this boundary, and to receive the mesh displacement calculations from the ANSYS solver under the effect of forces from CFX or other defined loads.

When an ANSYS Multi-field specification is being made in CFX-Pre, it is necessary to provide the name and number of the matching **Fluid Solid Interface** that was created in the Mechanical application, in the form of `FSIN_#`, where # is the interface number that was created in the Mechanical application. Since the interface number in the Mechanical application was 1, the name in question is `FSIN_1`. (If the interface number had been 2, then the name would have been `FSIN_2`, and so on.)

On this boundary, CFX will send ANSYS the forces on the interface, and ANSYS will send back the total mesh displacement it calculates given the forces passed from CFX and the other defined loads.

1. Create a new boundary named `Interface`.
2. Apply the following settings

Tab	Setting	Value
Basic Settings	Boundary Type	Wall
	Location	Interface
Boundary Details	Mesh Motion > Option	ANSYS MultiField
	Mesh Motion > Receive From ANSYS	Total Mesh Displacement
	Mesh Motion > ANSYS Interface	FSIN_1
	Mesh Motion > Send to ANSYS	Total Force

3. Click **OK**.

23.12.4.2. Symmetry Boundaries

Since a 2D representation of the flow field is being modeled (using a 3D mesh with one element thickness in the Z direction) symmetry boundaries will be created on the low and high Z 2D regions of the mesh.

1. Create a new boundary named `Sym1`.
2. Apply the following settings

Tab	Setting	Value
Basic Settings	Boundary Type	Symmetry
	Location	Sym1

3. Click **OK**.
4. Create a new boundary named `Sym2`.
5. Apply the following settings

Tab	Setting	Value
Basic Settings	Boundary Type	Symmetry
	Location	Sym2

6. Click **OK**.

23.12.5. Setting Initial Values

Since a transient simulation is being modeled, initial values are required for all variables.

1. Click *Global Initialization*  .
2. Apply the following settings:

Tab	Setting	Value
Global Settings	Initial Conditions > Cartesian Velocity Components > U	0 [m s^-1] [a]
	Initial Conditions > Cartesian Velocity Components > V	0 [m s^-1] [a]
	Initial Conditions > Cartesian Velocity Components > W	0 [m s^-1] [a]
	Initial Conditions > Static Pressure > Relative Pressure	0 [Pa] [a]

[a]These settings ensure that the fluid is at rest initially, and the flow is generated by the initial motion of the plate.

3. Click **OK**.

23.12.6. Setting Solver Control

Various ANSYS Multi-field settings are contained under **Solver Control** under the **External Coupling** tab. Most of these settings do not need to be changed for this simulation.

Within each timestep, a series of *coupling* or *stagger* iterations are performed to ensure that CFX-Solver, the Mechanical application and the data exchanged between the two solvers are all consistent. Within each stagger iteration, the Mechanical application and CFX-Solver both run once each, but which one runs first is a user-specifiable setting. In general, it is slightly more efficient to choose the solver that drives the simulation to run first. In this case, the simulation is being driven by the initial pressure applied in the Mechanical application, so the Mechanical application is set to solve before CFX-Solver within each stagger iteration.

1. Click *Solver Control*  .
2. Apply the following settings:

Tab	Setting	Value
Basic Settings	Transient Scheme > Option	Second Order Backward Euler
	Convergence Control > Max. Coeff. Loops	3
External Coupling	Coupling Step Control > Solution Sequence Control > Solve ANSYS Fields	Before CFX Fields
	Coupling Data Transfer Control > Ansys Variable	FZ
	Coupling Data Transfer Control > Ansys Variable > FZ	(Selected)
	Coupling Data Transfer Control > Ansys Variable > FZ > Convergence Target	(Selected)
	Coupling Data Transfer Control > Ansys Variable > FZ > Convergence Target > Convergence Target	1 [a]

Tab	Setting	Value
	Coupling Data Transfer Control > Ansys Variable	UZ
	Coupling Data Transfer Control > Ansys Variable > UZ	(Selected)
	Coupling Data Transfer Control > Ansys Variable > UZ > Convergence Target	(Selected)
	Coupling Data Transfer Control > Ansys Variable > UZ > Convergence Target > Convergence Target	1 [a]

[a]Since the Z component of both the force (FZ) and resultant displacement (UZ) are negligible for this 2-D case, their convergence targets are set to large values in order to negate their influence when determining load convergence.

3. Click **OK**.

23.12.7. Setting Output Control

This step sets up transient results files to be written at set intervals.

1. Click *Output Control* .
2. Click the **Trn Results** tab.
3. In the **Transient Results** tree view, click *Add new item* , accept the default name and click **OK**.
4. Apply the following settings:

Setting	Value
Option	Selected Variables
Output Variable List	Pressure, Total Mesh Displacement, Velocity
Output Frequency > Option	Every Coupling Step[a]

[a]This setting writes a transient results file every multi-field timestep.

5. Click the **Monitor** tab.
6. Select **Monitor Options**.
7. Under **Monitor Points and Expressions**:
   1. Click *Add new item* and accept the default name.
   2. Set **Option** to `Cartesian Coordinates`.
   3. Set **Output Variables List** to `Total Mesh Displacement X`.
   4. Set **Cartesian Coordinates** to `[0, 1, 0]`.

      This monitor point measures the x-component of the total mesh displacement at the top of the plate.
8. Click **OK**.

The settings for fluid physics are now complete. From the main menu, select **File > Save Project** to save these changes to the project, and select **File > Quit** to close ANSYS CFX-Pre and return to the Project Schematic.

23.13. Obtaining a Solution Using CFX-Solver Manager

The execution of an ANSYS Multi-field simulation requires both the CFX and ANSYS solvers to be running and communicating with each other. This section outlines the steps to launch both solvers and monitor the output using ANSYS CFX-Solver Manager.

1. On the Project Schematic, double-click the **Solution** cell in the Fluid system to launch the ANSYS CFX-Solver Manager application.

   ANSYS Workbench generates the CFX-Solver input file and passes it to ANSYS CFX-Solver Manager.

2. In ANSYS CFX-Solver Manager, ensure that **Define Run** dialog box is displayed.

   On the **Define Run** dialog box, **Solver Input File** is set automatically by ANSYS Workbench. The CFX-Solver input file contains settings for an ANSYS Multi-field simulation, thus **MultiField** tab appears on the Define Run dialog box.

3. On the **MultiField** tab, **ANSYS Input File** is set automatically by ANSYS Workbench.

4. On UNIX systems, you may need to manually specify where the ANSYS installation is if it is not in the default location. In this case, you must provide the path to the `v130/ansys` directory.

5. Click **Start Run**.

Note

On the **Run Definition** tab, the **Initialization Option** field is set to **Current Solution Data (if possible)**, its default setting. These runs use the results from any previous solution run as initial values for a subsequent update. This may not be desirable when restarting transient runs, which typically need to start from the initial conditions specified in the Setup cell. See Properties View in the *CFX Introduction* for more details.

The run begins by some initial processing of the ANSYS Multi-field input which results in the creation of a file containing the necessary multi-field commands for ANSYS, and then the ANSYS Solver is started. The CFX Solver is then started in such a way that it knows how to communicate with the ANSYS Solver.

After the run is under way, two new plots appear in ANSYS CFX-Solver Manager:

- **ANSYS Field Solver (Structural)** This plot is produced only when the solid physics is set to use large displacements or when other non-linear analyses are performed. It shows convergence of the ANSYS Solver. Full details of the quantities are described in the ANSYS user documentation. In general, the `CRIT` quantities are the convergence criteria for each relevant variable, and the `L2` quantities represent the `L2` Norm of the relevant variable. For convergence, the `L2` Norm should be below the criteria. The x-axis of the plot is the cumulative iteration number for ANSYS, which does not correspond to either timesteps or stagger iterations. Several ANSYS iterations will be performed for each timestep, depending on how quickly ANSYS converges. You will usually see a somewhat *spiky* plot, as each quantity will be unconverged at the start of each timestep, and then convergence will improve.
- **ANSYS Interface Loads (Structural)** This plot shows the convergence for each quantity that is part of the data exchanged between the CFX and ANSYS Solvers. Six lines appear, corresponding to three force components (FX, FY, and FZ) and three displacement components (UX, UY, and UZ). Each quantity is converged when the plot shows a negative value. The x-axis of the plot corresponds to the cumulative

number of stagger iterations (coupling iterations) and there are several of these for every timestep. Again, a spiky plot is expected as the quantities will not be converged at the start of a timestep.

The ANSYS out file is displayed in ANSYS CFX-Solver Manager as an extra tab. Similar to the CFX out file, this is a text file recording output from ANSYS as the solution progresses.

1. Click the **User Points** tab and watch how the top of the plate displaces as the solution develops.

   When the solver run has finished, a completion message appears in a dialog box.
2. Click **OK**.

From the main menu, select **File > Quit** to close ANSYS CFX-Solver Manager and return to the Project Schematic.

23.14. Viewing Results in ANSYS CFD-Post

On the Project Schematic, double-click the **Results** cell in the Fluid system to launch the ANSYS CFD-Post application.

Being an ANSYS Multi-field run, both the CFX and ANSYS results files will be opened up in CFD-Post.

23.14.1. Plotting Results on the Solid

When ANSYS CFD-Post reads an ANSYS results file, all the ANSYS variables are available to plot on the solid, including stresses and strains. The mesh regions available for plots by default are limited to the full boundary of the solid, plus certain named regions which are automatically created when particular types of load are added in Simulation. For example, any Fluid-Solid Interface will have a corresponding mesh region with a name such as `FSIN 1`. In this case, there is also a named region corresponding to the location of the fixed support, but in general pressure loads do not result in a named region.

You can add extra mesh regions for plotting by creating named selections in Simulation - see the Simulation product documentation for more details. Note that the named selection must have a name which contains only English letters, numbers and underscores for the named mesh region to be successfully created.

Note that when ANSYS CFD-Post loads an ANSYS results file, the true global range for each variable is not automatically calculated, as this would add a substantial amount of time depending on how long it takes to load such a file (you can turn on this calculation using **Edit** > **Options** and using the **Pre-calculate variable global ranges** setting under **CFD-Post** > **Files**). When the global range is first used for plotting a variable, it is calculated as the range within the current timestep. As subsequent timesteps are loaded into CFD-Post, the Global Range is extended each time variable values are found outside the previous Global Range.

1. Turn on the visibility of `Default Boundary` (under `ANSYS at 5s` > `Default Domain`).
2. Right-click a blank area in the viewer and select **Predefined Camera** > **View From +Z**. Zoom into the plate to see it clearly.
3. Apply the following settings to `Default Boundary`:

Tab	Setting	Value
Color	Mode	Variable
	Variable	Von Mises Stress

4. Click **Apply**.

5. Select **Tools** > **Timestep Selector** from the task bar to open the **Timestep Selector** dialog box. Notice that a separate list of timesteps is available for each results file loaded, although for this case the lists are the same. By default, **Sync Cases** is set to `By Time Value` which means that each time you change the timestep for one results file, CFD-Post will automatically load the results corresponding to the same time value for all other results files.
6. Set **Match** to **Nearest Available**.
7. Change to a time value of 1 [s] and click **Apply**.

The corresponding transient results are loaded and you can see the mesh move in both the CFX and ANSYS regions.

1. Turn off the visibility of `Default Boundary` (under `ANSYS at 1s` > `Default Domain`).
2. Create a contour plot, set **Locations** to ANSYS > `Default Boundary` and `Fluid` > `Sym2`, and set **Variable** to `Total Mesh Displacement`. Click **Apply**.
3. Using the timestep selector, load time value 0.8 [s] (which is where the maximum total mesh displacement occurs).

This verifies that the contours of `Total Mesh Displacement` are continuous through both the ANSYS and CFX regions.

Many FSI cases will have only relatively small mesh displacements, which can make visualization of the mesh displacement difficult. ANSYS CFD-Post allows you to visually magnify the mesh deformation for ease of viewing such displacements. Although it is not strictly necessary for this case, which has mesh displacements which are easily visible unmagnified, this is illustrated by the next few instructions.

1. Using the timestep selector, load time value 0.1 [s] (which has a much smaller mesh displacement than the currently loaded timestep).
2. Place the mouse over somewhere in the viewer where the background color is showing. Right-click and select **Deformation** > **Auto**. Notice that the mesh displacements are now exaggerated. The **Auto** setting is calculated to make the largest mesh displacement a fixed percentage of the domain size.
3. To return the deformations to their true scale, right-click and select **Deformation** > **True Scale**.

23.14.2. Creating an Animation

1. Using the **Timestep Selector** dialog box, ensure the time value of 0.1 [s] is loaded.
2. Turn off the visibility of `Contour 1`.
3. Turn on the visibility of `Sym2`.
4. Apply the following settings to `Sym2`.

Tab	Setting	Value
Color	Mode	Variable
	Variable	Pressure

5. Click **Apply**.
6. Create a vector plot, set **Locations** to `Sym1` and leave **Variable** set to `Velocity`. Set **Color** to be `Constant` and choose black. Click **Apply**.
7. Turn on the visibility of `Default Boundary` (under `ANSYS at 0.1s` > `Default Domain`), and set **Color** to a constant blue.

8. Click *Animation*  .

   The **Animation** dialog box appears.

9. Select **Keyframe Animation**.

10. In the **Animation** dialog box:

    1. Click *New*  to create `KeyframeNo1`.
    2. Highlight `KeyframeNo1`, then change **# of Frames** to 48.
    3. Load the last timestep (`50`) using the timestep selector.
    4. Click *New*  to create `KeyframeNo2`.

       The **# of Frames** parameter has no effect for the last keyframe, so leave it at the default value.
    5. Select **Save Movie**.
    6. Set **Format** to `MPEG1`.
    7. Click *Browse*  next to **Save Movie** to set a path and file name for the movie file.

       If the file path is not given, the file will be saved in the directory from which CFD-Post was launched.
    8. Click **Save**.

       The movie file name (including path) will be set, but the movie will not be created yet.
    9. If frame 1 is not loaded (shown in the **F:** text box in the middle of the **Animation** dialog box), click *To Beginning*  to load it.

       Wait for CFD-Post to finish loading the objects for this frame before proceeding.
    10. Click *Play the animation*  .

        The movie will be created as the animation proceeds. This will be slow, since a timestep must be loaded and objects must be created for each frame. To view the movie file, you need to use a viewer that supports the MPEG format.
    11. Save the results by selecting **File > Save Project** from the main menu.

When you are finished viewing results in ANSYS CFD-Post, return to the Project Schematic and select **File > Exit** to exit from ANSYS Workbench.

Chapter 24: Optimizing Flow in a Static Mixer

Design Exploration is a Workbench component that you can use to examine the effect of changing parameters in a system. In this example, you will see how changing the geometry and physics of a static mixer changes the effectiveness of the mixing of water at two different temperatures. The measure of the mixing effectiveness will be the output temperature range.

This tutorial includes:

Note

Some of the instructions in this tutorial assume that you have sufficient licensing to have multiple applications open. If you do not have sufficient licensing, you may not be able to keep as many of the applications open as this tutorial suggests. In this case, simply close the applications as you finish with them.

24.1. Tutorial Features

In this tutorial you will learn about:

- Creating a geometry in DesignModeler and creating a mesh.
- Using General mode in CFX-Pre to set up a problem.
- Using design points to manually vary characteristics of the problem to see how you can improve the mixing.
- Using Design Exploration to vary characteristics of the problem programmatically to find an optimal design.

Component	Feature	Details
DesignModeler		Geometry Creation
		Named Selections
Meshing Application		Mesh Creation
CFX-Pre	User Mode	General Mode

Component	Feature	Details
	Analysis Type	Steady State
	Fluid Type	General Fluid
	Domain Type	Single Domain
	Turbulence Model	k-Epsilon
	Heat Transfer	Thermal Energy
	Boundary Conditions	Inlet (Subsonic)
		Outlet (Subsonic)
		Wall: No-Slip
		Wall: Adiabatic
	Timescale	Physical Timescale
	Expressions	Workbench input parameter
CFD-Post	Expressions	Workbench output parameter
Parameters	Design Points	Manual changes
Design Exploration	Goal Driven Optimization	Design of Experiments
		Response Surface
		Optimization

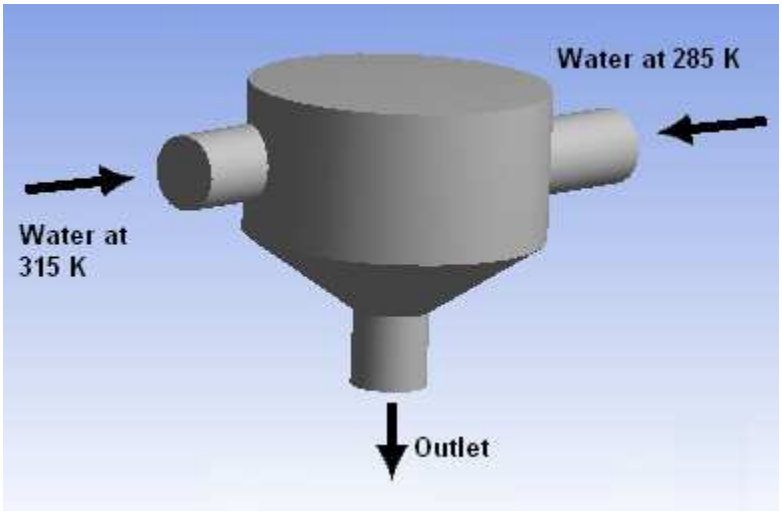
24.2. Overview of the Problem to Solve

This tutorial simulates a static mixer consisting of two inlet pipes delivering water into a mixing vessel; the water exits through an outlet pipe. A general workflow is established for analyzing the flow of fluid into and out of a mixer.

Initially, water enters through both pipes at the same rate but at different temperatures. The first inlet has a mass flow rate that has an initial value of 1500 kg/s and a temperature of 315 K. The second inlet also has a mass flow rate that has an initial value of 1500 kg/s, but at a temperature of 285 K. The radius of the mixer is 2 m.

Your goal in this tutorial is to understand how to use Design Points and Design Exploration to optimize the amount of mixing of the water when it exits the static mixer, as measured by the distribution of the water's temperature at the outlet.

Figure 24.1 Static Mixer with 2 Inlet Pipes and 1 Outlet Pipe



24.3. Setting Up ANSYS Workbench

Before you begin using ANSYS Workbench, you have to configure the **Geometry Import** option settings for use with this tutorial:

1. Launch ANSYS Workbench.
2. From the ANSYS Workbench menu bar, select **Tools** > **Options**. The **Options** configuration dialog appears.
3. In the **Options** configuration dialog, select **Geometry Import**
   a. Ensure **Parameters** is selected and remove the "DS" from **Filtering Prefixes and Suffixes**.
   b. Select **Named Selections** and remove the name "NS" from **Filtering Prefixes**.
   c. Click **OK**.

24.4. Creating the Project

To create the project, you save an empty project:

1. From the ANSYS Workbench menu bar, select **File** > **Save As** and save the project as `StaticMixer-DX.wbpj` in the directory of your choice.
2. From **Toolbox** > **Analysis Systems**, drag the **Fluid Flow (CFX)** system onto the **Project Schematic**.

24.5. Creating the Geometry in DesignModeler

Now you can create a geometry by using DesignModeler:

1. In the **Fluid Flow (CFX)** system, right-click **Geometry** and select **New Geometry**.

   DesignModeler starts.
2. If DesignModeler displays a dialog box for selecting the desired length unit, select **Meter** as the desired length unit and click **OK**.

   Note that this dialog box will not appear if you have previously set a default unit of measurement.

24.5.1. Creating the Solid

You create geometry in DesignModeler by creating two-dimensional sketches and extruding, revolving, sweeping, or lofting these to add or remove material. To create the main body of the static mixer, you will draw a sketch of a cross-section and revolve it.

1. In the **Tree Outline**, click **ZXPlane**.

   Each sketch is created in a plane. By selecting **ZXPlane** before creating a sketch, you ensure that the sketch you are about to create is based on the ZX plane.

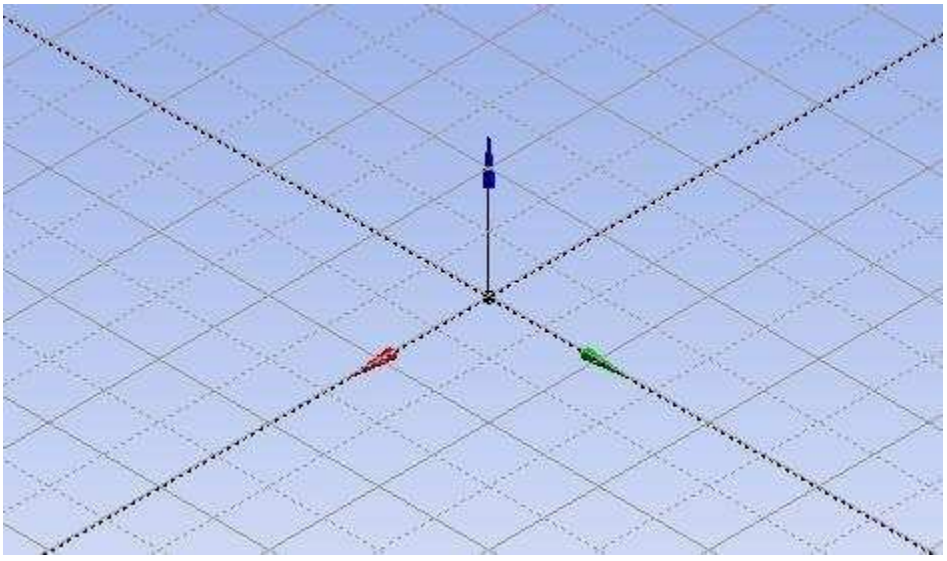
2. Click *New Sketch* on the Active Plane/Sketch toolbar, which is located above the Graphics window.
3. In the **Tree Outline**, click **Sketch1**.
4. Select the **Sketching** tab (below the **Tree Outline**) to view the available sketching toolboxes.

24.5.1.1. Setting Up the Grid

Before starting your sketch, set up a grid on the plane in which you will draw the sketch. The grid facilitates the precise positioning of points (when **Snap** is enabled).

1. Click **Settings** (in the **Sketching** tab) to open the **Settings** toolbox.
2. Click **Grid** and select **Show in 2D** and **Snap**.
3. Click **Major Grid Spacing** and set it to 1.
4. Click **Minor-Steps per Major** and set it to 2.
5. To see the effect of changing **Minor-Steps per Major**, press and hold the right mouse button above and to the left of the plane center in the Graphics window, then drag the mouse down and right to make a box around the plane center.

   When you release the mouse button, the model is magnified to show the selected area.



You now have a grid of squares with the smallest squares being 50 cm across. Because snap is enabled, you can select only points that are on this grid to build your geometry. Using snap can help you to position objects correctly.

The triad at the center of the grid indicates the local coordinate frame. The color of the arrow indicates the local axis: red for X, green for Y, and blue for Z.

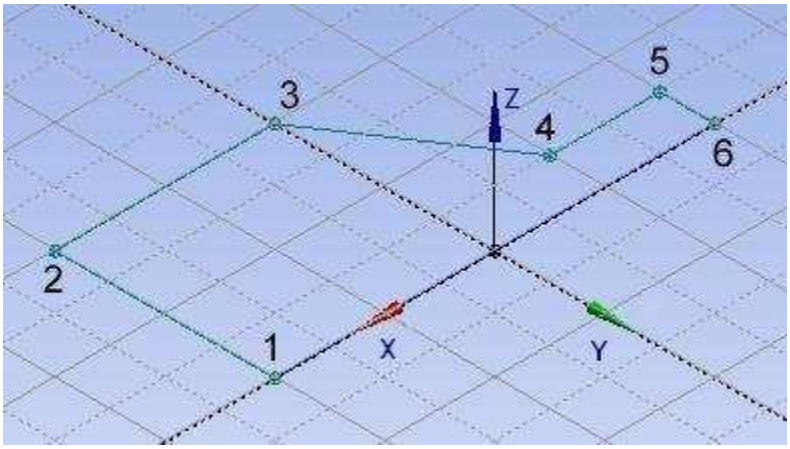
24.5.1.2. Creating the Basic Geometry

Start by creating the main body of the mixer:

1. From the **Sketching** tab, select the **Draw** toolbox.
2. Click **Polyline** and then create the shape shown below as follows:
   a. Click the grid in the position where one of the points from the shape needs to be placed (it does not matter which point, but a suggested order is given in the graphic below).
   b. Click each successive point to make the shape.

      If at any time you click the wrong place, right-click over the Graphics window and select **Back** from the shortcut menu to undo the last point selection.
   c. To close the polyline after selecting the last point, right-click and select **Closed End** from the shortcut menu.

Information about the new sketch, **Sketch1**, appears in the details view. Note that the longest straight line (4 m long) in the diagram below is along the local X-axis (located at Y = 0 m). The numbers and letters in the image below are added here for your convenience but do not appear in the software.



24.5.1.3. Revolving the Sketch

You will now create the main body of the mixer by revolving the new sketch around the local X-axis.

1. Click *Revolve* from the toolbar above the Graphics window.

   Details of the Revolve feature are shown in the **Details View** at the bottom left of the window.
2. Leave the name of the Revolve feature at its default value: **Revolve1**.
3. Leave **Base Object** set to **Sketch1**.

   The **Base Object** specifies which sketch is to be revolved.
4. In the **Details View** you should see **Apply** and **Cancel** buttons next to the **Axis** property; if those buttons are not displayed, click the word **Axis**.

5. In the Graphics window, click the grid line that is aligned with the local X-axis (the local X-axis, represented by a red arrow, is parallel to the global Z-axis in this case), then click **Apply** in the **Details View**.

   The text next to **Axis** changes to **Selected**.

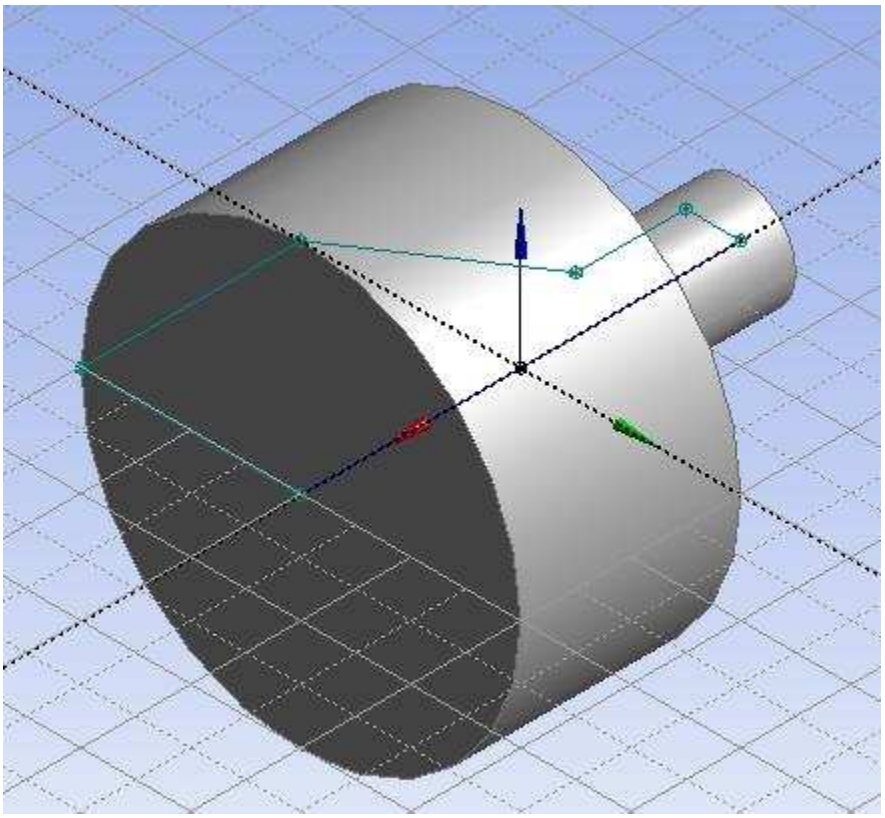
6. Leave **Operation** set to **Add Material** because you need to create a solid (which will eventually represent a fluid region).

7. Ensure that **Angle** is set to **360°** and leave the other settings at their defaults.

8. Click *Generate*  to activate the Revolve operation.

   You can select this from the 3D Features Toolbar, from the shortcut menu by right-clicking in the Graphics window, or from the shortcut menu by right-clicking the **Revolve1** object in the **Tree Outline**.

After generation, you should find that you have a solid as shown below.



24.5.1.4. Create the First Inlet Pipe

To create the inlet pipes, you will create two sketches and extrude them. For clear viewing of the grid during sketching, you will hide the previously created geometry.

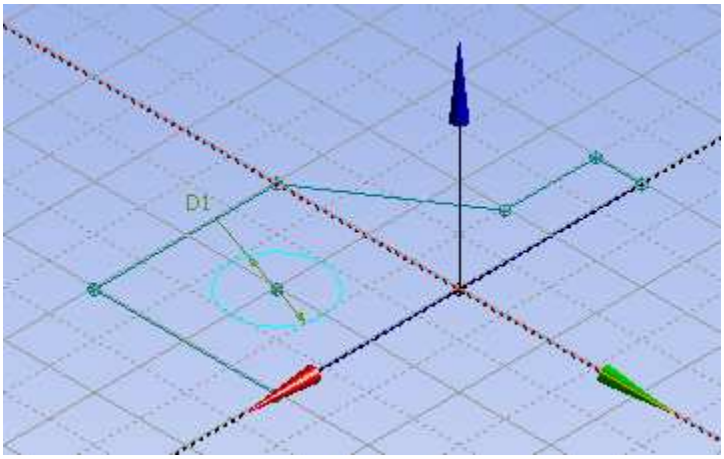
1. In the **Tree Outline**, click the plus sign next to **1 Part, 1 Body** to expand the tree structure.
2. Right-click **Solid** and select **Hide Body**.
3. Select **ZXPlane** in the **Tree Outline**.
4. Click *New Sketch*  .
5. From the **Sketching** tab, select the **Draw** toolbox.
6. Click **Circle** and then create the circle shown below as follows:

a. Click and hold the left mouse button at the center of the circle.

b. While still holding the mouse button, drag the mouse to set the radius.

c. Release the mouse button.

7. Select the **Dimensions** toolbox, select **General**, click the circle in the sketch, then click near the circle to set a dimension. In the Details View, select the check box beside **D1**. When prompted, rename the parameter to `inDia` and click **OK**. This dimension will be a parameter that is modified in Design Exploration.



24.5.1.4.1. Extrude the First Side-pipe

To create the first side-pipe extrude the sketch:

1. Click *Extrude* from the 3D Features toolbar, located above the Graphics window.
2. In the **Details View**, change **Direction** to **Reversed** to reverse the direction of the extrusion (that is, click the word **Normal**, then from the drop-down menu select **Reversed**).
3. Change **Depth** to **3** (meters) and press **Enter** to set this value.

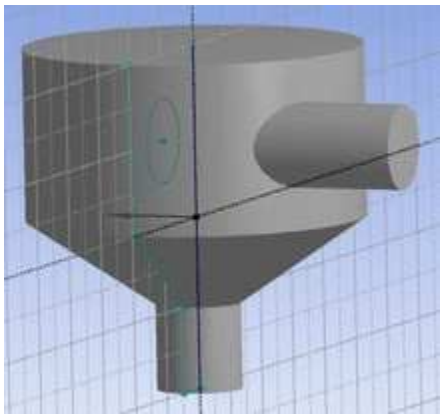
   All other settings should remain at their default values. Note that the **Operation** property is set to **Add Material**, which indicates that material is to be added to the existing solid.
4. Click *Generate* to perform the extrusion.

   Initially, you will not see the geometry.

24.5.1.4.2. Make the Solid Visible

To see the result of the previous operation, make the solid visible:

1. In the **Tree Outline**, right-click **Solid** and select **Show Body**.
2. Click and hold the middle mouse button over the middle of the Graphics window and drag the mouse to rotate the model. The solid should be similar to the one shown below.



3. Right-click **Solid** and select **Hide Body**.

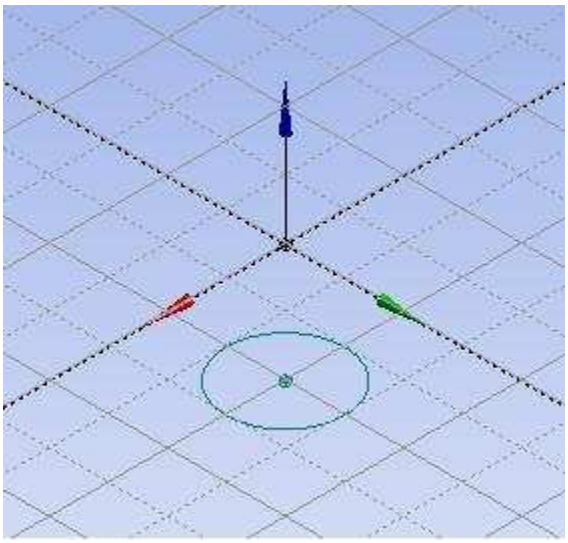
24.5.1.5. Create the Second Inlet Pipe

You will create the second inlet so that the relative angle between the two inlets is controlled parametrically, enabling you to evaluate the effects of different relative inlet angles:

1. In the **Tree Outline**, select **ZXPlane**.
2. In the toolbar, click *New Plane* .

   The new plane (**Plane4**) appears in the **Tree Outline**.
3. In the **Details View**, click beside **Transform 1 (RMB)** and choose the axis about which you want to rotate the inlet: **Rotate about X**.
4. Select the check box for the **FD1, Value 1** property then, when prompted, set the name to `in2Angle` and click **OK**.

   This makes the angle of rotation of this plane a design parameter.
5. Click *Generate* .
6. In the **Tree Outline** click **Plane4**.
7. Create a new sketch (**Sketch3**) based on **Plane4** by clicking *New Sketch* .
8. Select the **Sketching** tab.
9. Click **Settings** to open the **Settings** toolbox.
10. Click **Grid** and select **Show in 2D** and **Snap**.
11. Click **Major Grid Spacing** and set it to **1**.
12. Click **Minor-Steps per Major** and set it to **2**.
13. Right-click over the Graphics window and select **Isometric View** to put the sketch into a sensible viewing position.
14. Zoom in, if required, to see the level of detail in the image below.
15. From the **Draw** Toolbox, select **Circle** and create a circle as shown below:



16. Select the **Dimensions** toolbox, click **General**, click the circle in the sketch, then click near the circle to set a dimension.
17. In the **Details View**, select the check box for the **D1** property then, when prompted, set the name to `inDia` and click **OK**.
18. Click *Extrude*  .
19. In the **Details View**, ensure **Direction** is set to **Normal** in order to extrude in the same direction as the plane normal.
20. Ensure that **Depth** is set to **3** (meters).

    All other settings should remain at their default values.
21. Click *Generate*  to perform the extrusion.
22. Right-click **Solid** in the **Tree Outline** and select **Show Body**.

    The geometry is now complete.

24.5.1.6. Create Named Selections

Named selections enable you to specify and control like-grouped items. Here, you will create named selections so that you can specify boundary conditions in CFX-Pre for these specific regions.

Note

The Graphics window must be in “viewing mode” for you to be able to orient the geometry and the Graphics window must be in "select mode" for you to be able to select a boundary in the geometry. You set viewing mode or select mode by clicking the icons in the toolbar:

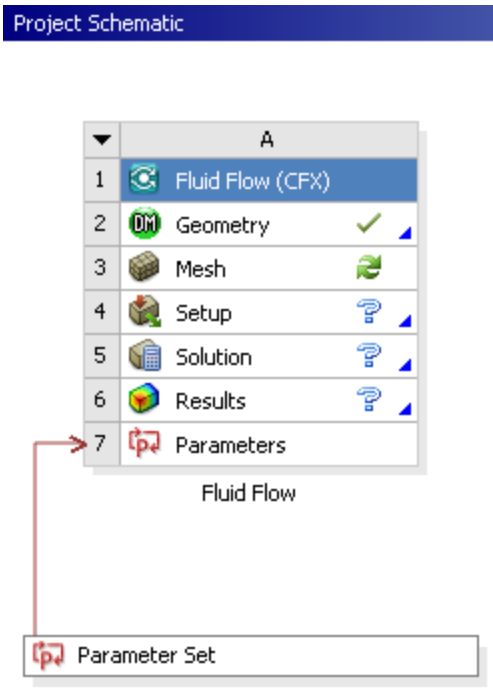
Create named selections as follows:

1. In viewing mode, orient the static mixer so that you can see the inlet that has the lowest value of (global) Y-coordinate.

   You can rotate the mixer by holding down the middle-mouse button (or the mouse scroll wheel) while moving the mouse.

2. In select mode, with *Selection Filter: Model Faces (3D)*  active (in the toolbar), click the inlet face to select it, then right-click the inlet and select **Named Selection**.
3. In the **Details View**, click **Apply**.
4. Set **Named Selection** to `in1`.
5. Click *Generate*  .
6. In viewing mode, orient the static mixer so that you can see the inlet that has the highest value of (global) Y-coordinate.
7. In select mode, click the inlet face to select it, then right-click the inlet and select **Named Selection**.
8. In the **Details View**, click **Apply**.
9. Set **Named Selection** to `in2`.
10. Click *Generate*  .
11. In viewing mode, orient the static mixer so that you can see the outlet (the face with the lowest value of (global) Z-coordinate).
12. In select mode, click the outlet face to select it, then right-click the outlet and select **Named Selection**.
13. In the **Details View**, click **Apply**.
14. Set **Named Selection** to `out`.
15. Click *Generate*  .
16. Click **Save** on the ANSYS Workbench toolbar.

    This enables you to recover the work that you have performed to this point if needed (until the next time you save the tutorial).



24.6. Creating the Mesh

To create the mesh:

1. In the **Project Schematic**, right-click the **Mesh** cell and select **Edit**.

   The Meshing application appears.
2. Right-click **Project** > **Model (A3)** > **Mesh** and select **Generate Mesh**.
3. After the mesh has been produced, return to the **Project Schematic**, right-click the **Mesh** cell, and select **Update**.
4. In the Meshing application, select **File** > **Close Meshing**.

24.7. Setting up the Case with CFX-Pre

Now that the mesh has been created, you can use CFX-Pre to define the simulation. To set up the case with CFX-Pre:

1. Double-click the **Setup** cell. CFX-Pre appears with the mesh file loaded.
2. In CFX-Pre, create an expression named `inMassFlow`:

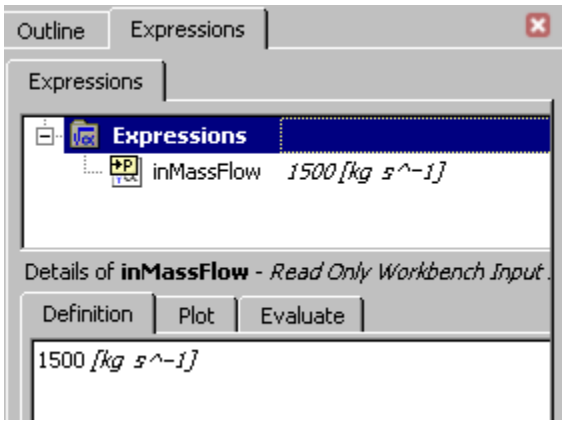
   a. In the **Outline** tree view, expand **Expressions, Functions and Variables** and right-click **Expression** and select **Insert** > **Expression**.

   b. Give the new expression the name: `inMassFlow`

   c. In the **Definition** area, type: `1500 [kg s^-1]`

   d. Click **Apply**.

3. Right-click **inMassFlow** in the **Expressions** area and select **Use as Workbench Input Parameter**. A small "P" with a right-pointing arrow appears on the expression's icon.



4. Define the characteristics of the domain:

   a. Click the **Outline** tab.

   b. Double-click **Simulation** > **Flow Analysis 1** > **Default Domain** to open it for editing.

   c. Apply the following settings:

Tab	Setting	Value
Basic Settings	Fluid and Particle Definitions	Fluid 1
	Fluid and Particle Definitions > Fluid 1 > Material	Water
	Domain Models > Pressure > Reference Pressure	1 [atm]
Fluids Models	Heat Transfer > Option	Thermal Energy

   d. Click **OK**.

5. Create the first inlet boundary:

   a. From the CFX-Pre menu bar, select **Insert** > **Boundary**.

   b. In the **Insert Boundary** dialog, name the new boundary `in1` and click **OK**.

   c. Apply the following settings to `in1`:

Tab	Setting	Value
Basic Settings	Boundary Type	Inlet
	Location	in1
Boundary Details	Mass and Momentum > Option	Mass Flow Rate
	Mass Flow Rate	inMassFlow [1] (p. 434)
	Heat Transfer > Static Temperature	315 [K]
1. To enter this expression name into the Mass Flow Rate field, click in the blank field, click the *Enter Expression* icon that appears, right-click in the blank field, then select the inMassFlow expression that appears.		

d. Click **OK**.

6. Create the second inlet boundary:

   a. From the CFX-Pre menu bar, select **Insert** > **Boundary**.

   b. In the **Insert Boundary** dialog, name the new boundary `in2` and click **OK**.

   c. Apply the following settings to `in2`:

Tab	Setting	Value
Basic Settings	Boundary Type	Inlet
	Location	in2
Boundary Details	Mass and Momentum > Option	Mass Flow Rate
	Mass Flow Rate	inMassFlow [1 (p. 435)]
	Heat Transfer > Static Temperature	285 [K]
1. To enter this expression name into the Mass Flow Rate field, click in the blank field, click the *Enter Expression*  icon that appears, right-click in the blank field, then select the inMassFlow expression that appears.		

   d. Click **OK**.

7. Create the outlet boundary:

   a. From the CFX-Pre menu bar, select **Insert** > **Boundary**.

   b. In the **Insert Boundary** dialog, name the new boundary `out` and click **OK**.

   c. Apply the following settings to `out`:

Tab	Setting	Value
Basic Settings	Boundary Type	Outlet
	Location	out
Boundary Details	Mass and Momentum > Option	Static Pressure
Boundary Details	Mass and Momentum > Relative Pressure	0 [Pa]

   d. Click **OK**.

   CFX-Pre and ANSYS Workbench both update automatically. The three boundary conditions are displayed in the viewer as sets of arrows at the boundary surfaces. Inlet boundary arrows are directed into the domain; outlet boundary arrows are directed out of the domain.

8. Solver Control parameters control aspects of the numerical solution generation process. Set the solver controls as follows:

   a. Click *Solver Control*  .

   b. On the **Basic Settings** tab, set **Advection Scheme** > **Option** to `Upwind`.

While an upwind advection scheme is less accurate than other advection schemes, it is also more robust. This advection scheme is suitable for obtaining an initial set of results, but in general should not be used to obtain final results.

c. Set **Convergence Control** > **Min. Iterations** to 5.

This change is required because when the solver is restarted from a previous "converged" solution for each design point, the solver may "think" the solution is converged after one or two iterations and halt the solution prematurely if the default setting (1) is maintained.

d. Set **Convergence Control** > **Fluid Timescale Control** > **Timescale Control** to `Physical Timescale` and set the physical timescale value to 2 [s].

The time scale can be calculated automatically by the solver or set manually. The `Automatic` option tends to be conservative, leading to reliable, but often slow, convergence. It is often possible to accelerate convergence by applying a time scale factor or by choosing a manual value that is more aggressive than the `Automatic` option. By selecting a physical time scale, you obtain a convergence that is at least twice as fast as the `Automatic` option.

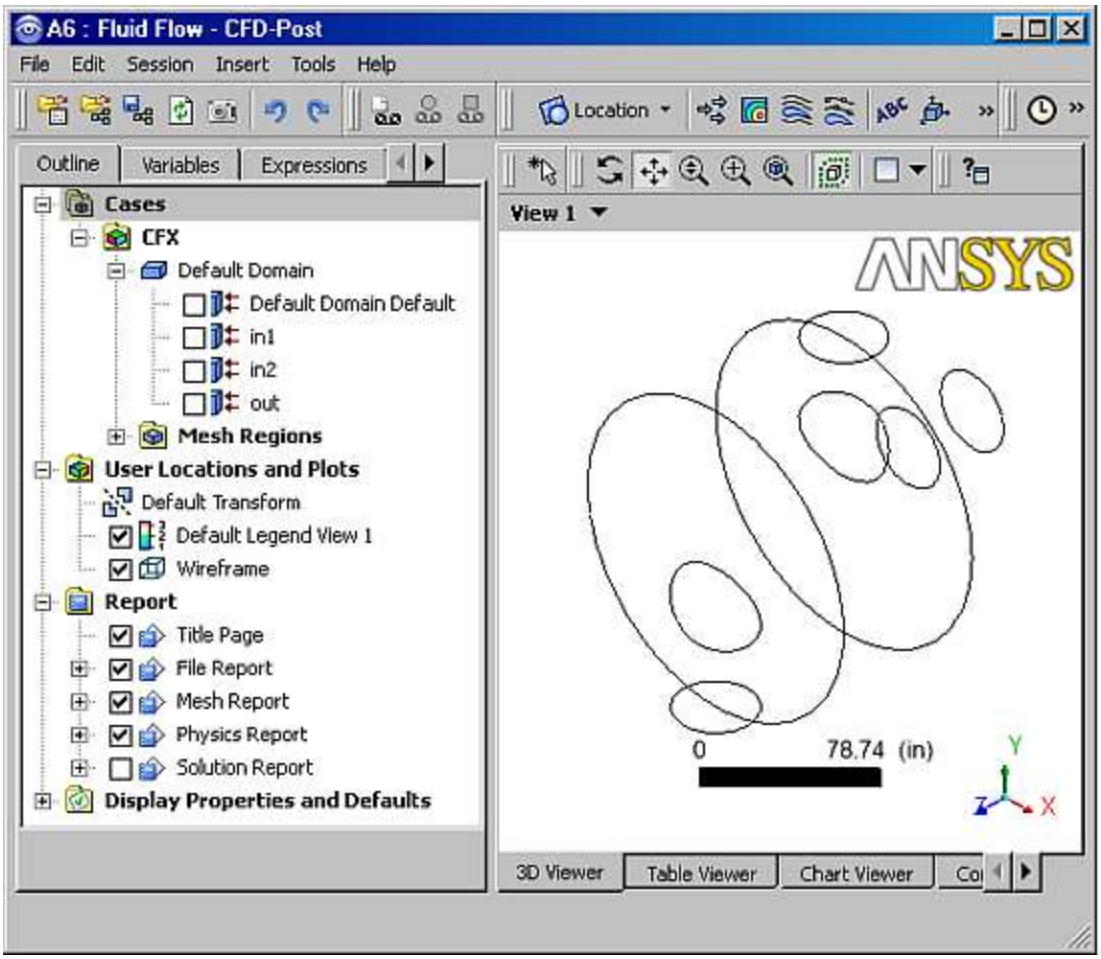
e. Click **OK**.

CFX-Pre and ANSYS Workbench both update automatically.

9. In the **Project Schematic**, right-click the **Solution** cell and select **Update**. CFX-Solver obtains a solution.
10. When the **Solution** cell shows an up-to-date state, right-click the **Results** cell and select **Refresh**. When the refresh is complete, right-click the **Results** cell again and select **Edit**. CFD-Post starts.

24.8. Setting the Output Parameter in CFD-Post

When CFD-Post starts, it displays the 3D Viewer and the **Outline** workspace.



You need to create an expression for the response parameter to be examined (Outlet Temperature) called `OutTempRange`, which will be the maximum output temperature minus the minimum output temperature:

1. On the **Expressions** tab, right-click **Expressions** > **New**.
2. Type `OutTempRange` and click **OK**.
3. In the **Definition** area:
   a. Right-click **Functions** > **CFD-Post** > **maxVal**.
   b. With the cursor between the parentheses, right-click and select **Variables** > **Temperature**.
   c. Left-click after the @, then right-click and select **Locations** > **out**.

      That specifies the maximum output temperature.
   d. Now, complete the expression so that it appears as follows:

      ```
      maxVal(Temperature)@out - minVal(Temperature)@out
      ```

   e. Click **Apply**.

      The new expression appears in the **Expressions** list. Note the value of the expression.
4. In the **Expressions** list, right-click **OutTempRange** and select **Use as Workbench Output Parameter**. A small "P" with a right-pointing arrow appears on the expression's icon.

5. Repeat the steps above for a second expression called `OutTempAve`. This expression will be used to monitor the output temperature. We expect the overall output temperature to be the average of the two input temperatures given that the incoming mass flows are equal. Make this expression's definition:

   ```
   massFlowAve(Temperature)@out
   ```

   Be sure to also set this expression to **Use as Workbench Output Parameter**. When you click **Apply** note the value of the expression.
6. Click **Save** on the ANSYS Workbench toolbar to save the project.
7. In CFD-Post, select **File** > **Quit**.

24.9. Investigating the Impact of Changing Design Parameters Manually

Now you will manually change the values of some design parameters to see what effect each has on the rate of mixing. These combinations of parameter values where you perform calculations are called *design points*. As you make changes to parameters in ANSYS Workbench, CFX-Pre and ANSYS DesignModeler will reflect the current value automatically; ensure that those programs are open so that you can see the changes take place. In particular, ensure that CFX-Pre has the **Expressions** view open.

1. In the **Project Schematic**, right-click **Parameters** (cell A7) and select **Edit**. A new set of views opens.
2. Resize the ANSYS Workbench window to be larger, then select **View** > **Project Schematic**. The **Project Schematic** reappears.

Tip

If necessary, you can close the **Toolbox** view to gain more space. To restore it, select **View** > **Toolbox**.

When you highlight Parameters (cell A7), among the new views are:

- **Outline of Schematic A7: Parameters**, which lists the input and output parameters and their values (which match the values observed in previous steps)
- **Table of Design Points**, which lists one design point named **Current**. Ensure that this view is wide enough to display the **Exported** column.

Now you will change the design parameter values from the **Outline of Schematic A7: Parameters** view:

1. In the **Project Schematic**, highlight **Parameters** (cell A7), then in the **Outline of Schematic A7: Parameters** view, change the **in2Angle** value from `0` to `-45` and press **Enter**.
2. In the **Project Schematic**, right-click **Geometry** and select **Update**. Notice how the geometry changes in ANSYS DesignModeler.
3. In the **Project Schematic**, highlight **Parameters** (cell A7), then in the **Outline of Schematic A7** view, change the **in2Angle** value from `-45` back to `0` and press **Enter**.
4. In the **Project Schematic**, right-click **Geometry** and select **Update**. Again, notice how the geometry changes in ANSYS DesignModeler.
5. In the **Project Schematic**, highlight **Parameters** (cell A7), then in the **Outline of Schematic A7** view, change the **inMassFlow** value from `1500` to `1600` and press **Enter**. Notice how the value of the expression has changed in the **Expressions** tree view in CFX-Pre; (the change is not reflected in the **Expressions** *Details* view unless you refresh the contents of the tab, for example by hiding and reopening it).

6. In the **Project Schematic**, highlight **Parameters** (cell A7), then in the **Outline of Schematic A7** view, change the **inMassFlow** value from `1600` back to `1500` and press **Enter**.

You have modified design parameter values and returned each to its original value. In doing this, the **Outline of Schematic** view's values and the **Table of Design Points** view's values have become out-of date. Right-click any cell in the **Table of Design Points** view's **Current** row and select **Update Selected Design Point**. This process updates the project and all of the ANSYS Workbench views. ANSYS Workbench may also close any open ANSYS CFX applications and run them in the background. When the update is complete, all of the results cells show current values and all of the cells that display status are marked as being up-to-date.

Now, you will make changes to design parameters as design points. You will create three design points, each of which will change the value of one parameter:

1. In the **Project Schematic**, highlight **Parameters** (cell A7). In the **Table of Design Points** view in the line under **Current**, make the following entries to create the first design point (**DP 1**). Notice that cells autofill with the values from the **Current** row, so you need enter only the value that differs from that:

   - **P1 – inDia**: `1`
   - **P2 – in2Angle**: `-45`
   - **P3 – inMassFlow**: `1500`
   - In the **Table of Design Points** > **Exported** column, select the check box.

   Note

   You should save the project once before you export a design point.

   Right-click in the row for **DP 1** and select **Update Selected Design Point**.

   ANSYS Workbench recalculates all of the values for the input and output parameters. All of the views are updated.

   Because you selected the check box in the **Exported** column, the update process writes a copy of the project (as *`project_name`*`_dp`*`dp_number`*`.wbpj`) so that you can refer back to the data for that design point.

2. Modify the design point (**DP 1**) using these values, including exporting the design point:

   - **P1 – inDia**: `1.5`
   - **P2 – in2Angle**: `0`
   - **P3 – inMassFlow**: `1500`

   Right-click in the row for **DP 1** and select **Update Selected Design Point**.

   If you had not kept the check box in the **Exported** column selected, the data in the design point's project file would not be rewritten and so the data in that file would not be consistent with the updated results now shown in ANSYS Workbench.

3. Repeat the previous step to create the second design point (**DP 2**) using these values:

   - **P1 – inDia**: `1`
   - **P2 – in2Angle**: `0`
   - **P3 – inMassFlow**: `1600`

Right-click in the row for **DP 2** and from the toolbar select **Update All Design Points**. (This command updates any out-of-date design points in a sequential fashion. In this case as only one design point is out of date, only it will be updated.)

4. Click **Save** on the ANSYS Workbench toolbar to save the project.

Recall that the goal of this design study is to maximize the mixing (which occurs when OutTempRange reaches its minimum value). From these manual tests, it appears that the best results are obtained by changing the input angle of one inlet. In all studies, the OutTempAve value stays very near a constant 300 K, as expected.

In the next section you will automate that manual process of repeatedly changing variable values by using **Design Exploration**.

24.10. Using Design of Experiments

In this section you will use **Design Exploration**'s Goal Driven Optimization feature to minimize the value of OutTempRange.

1. If you need to restore the **Toolbox**, select **View** > **Toolbox**. If no systems appear in the **Toolbox**, select a cell in the **Project Schematic** to refresh ANSYS Workbench.
2. From the **Design Exploration** toolbox, drag a **Goal Driven Optimization** system to the **Project Schematic** (under the **Parameter Set** bar).
3. Double-click the **Design of Experiments** cell.
4. In the **Outline of Schematic B2: Design of Experiments** view:

   a. Enable **P2 – in2Angle** (cell A6) and **P3 – inMassFlow** (cell A7); clear **P1 – inDia**.

   b. Select cell **P3 – inMassFlow** (cell A7). In the **Properties of Outline A6: P3** view, set:

      - **Lower Bound:** 1000
      - **Upper Bound:** 2000

   c. Select cell **P2 – in2Angle** (cell A6). In the **Properties of Outline A6: P2** view, set:

      - **Lower Bound:** –45
      - **Upper Bound:** 0

5. In the ANSYS Workbench toolbar, ensure that **View** > **Table** is set.
6. In the ANSYS Workbench toolbar, click **Preview Design of Experiments**. The **Table of Schematic B2: Design of Experiments** appears. This table has nine entries in the **Name** column, each of which represents a solver run to be performed. Beside the **Name** column are columns that have the values for the two input parameters, and a column to hold the value the solver will obtain for the output parameter. This preview gives an indication of the time that the nine solver runs will require.
7. In the **Project Schematic**, right-click the **Design of Experiments** cell and select **Update**. You can monitor the progress of the solver runs by clicking **Show Progress** in the lower-right corner of the ANSYS Workbench window.

   When the processing is complete, the **Table of Schematic B2: Design of Experiments** displays the results. Click the down-arrow on the **P4 – OutTempRange (K)** cell to sort in ascending order and show the best combination of input angle and mass flow. Note that the best results are returned from a low mass flow and the greatest difference in input angle.

24.11. Viewing the Response Surface

To view the response surface for this experiment:

1. In the ANSYS Workbench menu bar, ensure **View** > **Chart** is enabled.
2. In the **Project Schematic**, right-click **Response Surface** and select **Update**.
3. In the **Outline of Schematic B3; Response Surface** view, select **Response Surface** > **Response Points** > **Response Point** > **Response**. The results appear in various views:
   - The **Toolbox** shows the types of charts that are available.
   - The **Response Chart for P4 - OutTempRange** shows a 2D graph comparing OutTempRange to in2Angle.
   - The **Properties of Outline A12: Response** shows the values that are being used to display the 2D graph. Note that the other variable that was considered in the Design of Experiments (**inMassFlow**) is held at 1500.
4. Add the full range of **inMassFlow** results to the chart: right-click the chart and select **Toggle 2D/3D Chart**. A 3D chart appears that shows in more detail that the best results are returned from a low mass flow and the greatest difference in input angle.

24.12. Viewing the Optimization

To view the optimization for this experiment:

1. In the **Project Schematic**, highlight **Optimization**.
2. In the **Table of Schematic B4: Optimization**, change the **P4 - OutTempRange** / **Objective** cell from **No Objective** to **Minimize**.
3. In the **Project Schematic**, right-click **Optimization** and select **Update**.
4. In the **Table of Schematic B4: Optimization**, three design point candidates appear with their interpolated values. Recall that design points are combinations of parameter values where you perform real calculations (rather than relying on the interpolated values that appear in the response charts). To see all of the candidates represented graphically, in the **Outline of Schematic B4: Optimization**, select **Chart** > **Samples**. The **Samples Chart** appears with the three candidate samples highlighted.
5. In the **Table of Schematic B4: Optimization**, right-click the Candidate A value and select **Insert as Design Point(s)**.
6. In the **Project Schematic**, highlight **Parameters**. The various views update and in the **Table of Design Points** a new design point has appeared.
7. In the **Table of Design Points**, the new design point (**DP 3**) has no values and requests an update. Right-click the lightning icon and select **Update Selected Design Point**. You can follow the progress of the update in the **Project Schematic** as ANSYS Workbench reruns its calculations for the design point's parameters. Compare the **P4 - OutTempRange** value to the value given by the Design of Experiments calculations.

Chapter 25: Aerodynamic and Structural Performance of a Centrifugal Compressor

This tutorial includes:

Note

This tutorial is intended to be run on Windows-based machines only.

25.1. Tutorial Features

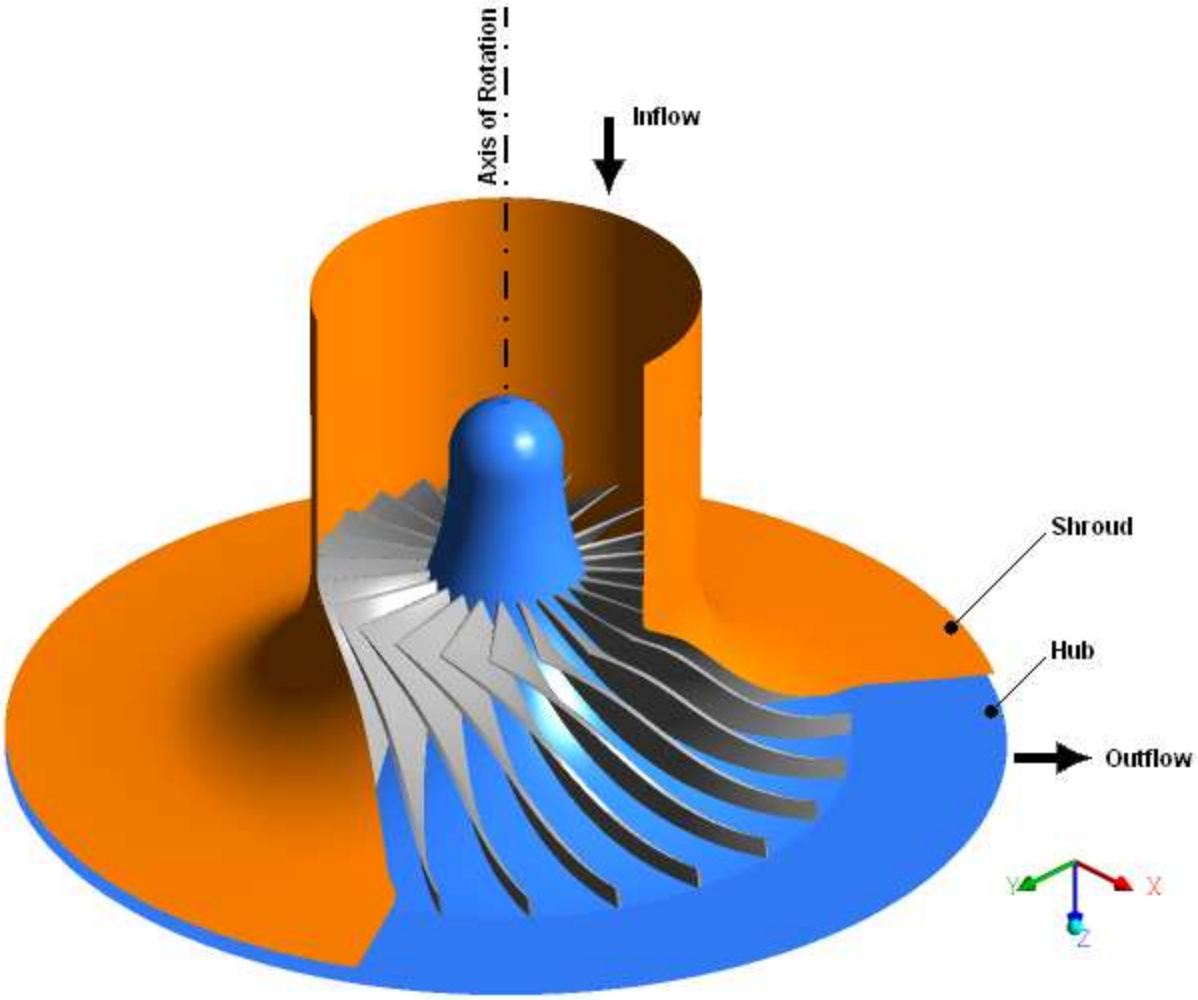
This tutorial addresses the following features of ANSYS Workbench.

Component	Feature	Details
ANSYS BladeGen	Geometry	Transfer of geometry to ANSYS TurboGrid.
ANSYS TurboGrid	Mesh	H/J/C/L-Grid Topology Shroud Tip defined by Profile Control Point Movements Edge Split Controls Transfer of Mesh to a CFX System
CFX-Pre	Turbo Mode	Used to define a Turbomachinery CFD Simulation
	Machine Type	Centrifugal Compressor
	Component Type	Rotating
	Analysis Type	Steady State

Component	Feature	Details
	Boundary Template	P-Total Inlet Mass Flow Outlet
	Flow Direction	Cylindrical Components
	Domain Type	Single Domain
	Timestep	Physical Timescale
CFD-Post	Report	Computed Results Table
		Blade Loading Span 50
		Streamwise Plot of Pt and P
		Velocity Streamlines Stream Blade TE
Static Structural System	Static Structural Analysis	CFX Pressure data applied to Structural Simulation
		Fixed Support
	Static Structural Solutions	Von Mises Stresses
		Total Deformation

25.2. Overview of the Problem to Solve

This tutorial makes use of several ANSYS software components to simulate the aerodynamic and structural performance of a centrifugal compressor.



The compressor has 24 blades that revolve about the Z-axis at 22360 RPM. A clearance gap exists between the blades and the shroud of the compressor. The outer diameter of the blade row is approximately 40 cm.

To begin analysis of the aerodynamic performance, a mesh will be created in ANSYS TurboGrid using an existing design which is to be reviewed beforehand in BladeGen. Once the mesh has been created, initial parameters defining the aerodynamic simulation will be set in CFX-Pre and then solved in CFX-Solver. The aerodynamic solution from the solver will then be processed and displayed in CFD-Post.

You will then use the Mechanical application to simulate structural stresses on the blade due to pressure loads from the aerodynamic analysis and rotationally-induced inertial effects. You will view an animation that shows the resulting blade distortion.

25.3. Before You Begin

If this is the first tutorial you are working with, it is important to review the following topics before beginning:

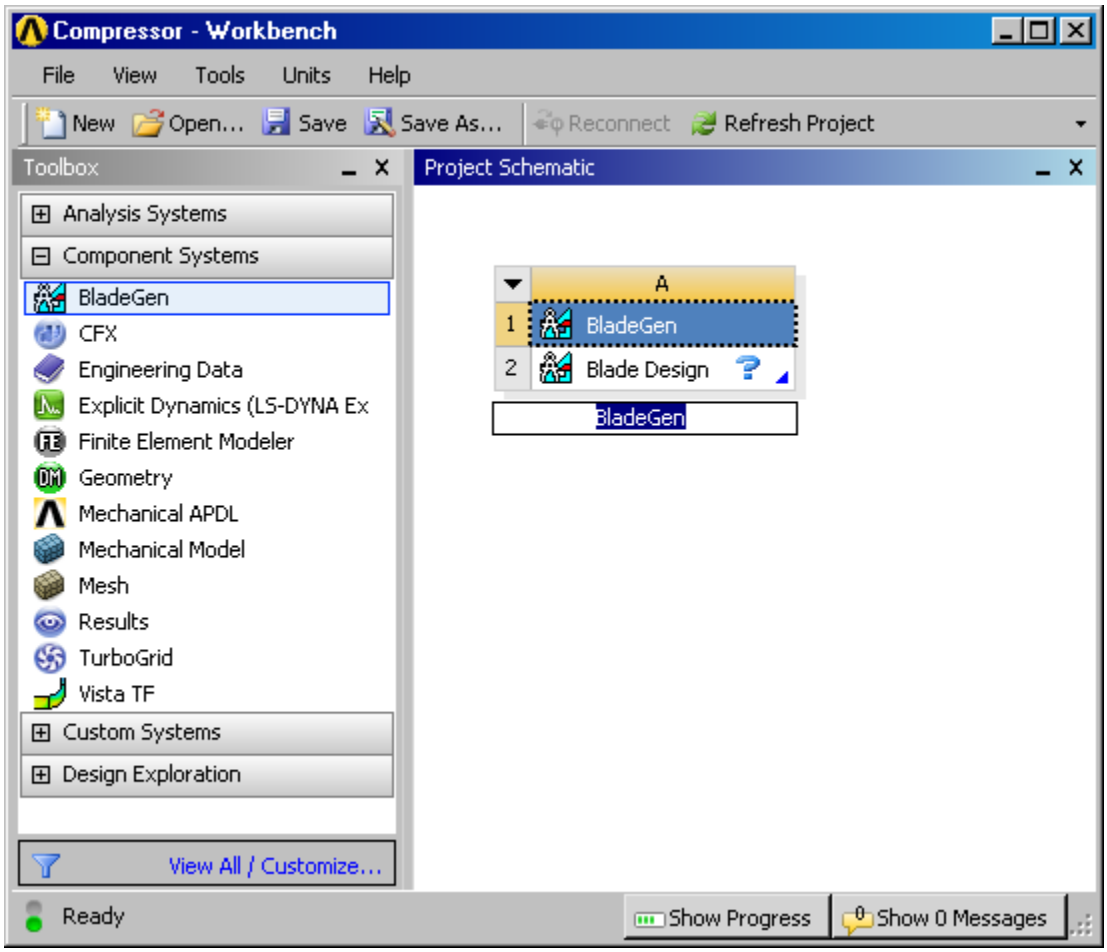
- *Running ANSYS CFX Tutorials in ANSYS Workbench* (p. 2)
- *Changing the Display Colors* (p. 5)

25.4. Starting ANSYS Workbench and Creating a BladeGen System

1. Use your operating system's tools to create a directory for your project's files. The directory you create will be referred to here as the *working directory*.

2. Copy the provided geometry file, `Centrifugal_Compressor.bgd`, from the `examples` directory to the working directory.
3. Start ANSYS Workbench.

   To do this in Microsoft Windows, from the **Start** menu select **All Programs > ANSYS 13.0 > Workbench**.
4. From the tool bar, click Save As... .
5. In the dialog box that appears, browse to the working directory, set **File name** to `Compressor`, and click **Save**.
6. In the **Toolbox** view, open **Component Systems** and double-click **BladeGen**. A **BladeGen** system opens in the **Project Schematic** view, and is ready to be given a name.



7. Type `Compressor` in the name field and then either press **Enter** or click outside the name field in order to end the rename operation.

   If you need to begin a new rename operation, right-click the blue **BladeGen** cell (cell A1) and select **Rename** from the shortcut menu.

   Now that renaming systems has been demonstrated, most of the other systems involved in this tutorial will simply use default names.
8. Later in this tutorial, you will require the **Blade Design** cell to provide the blade geometry to a **Geometry** cell of another system. To avoid the fluid zone surrounding the blade from also being provided, change the **Blade Design** cell properties as follows:

1. Right-click the **Blade Design** cell of the **BladeGen** system and select **Properties**.

   The **Properties** view shows properties that control how the geometry is imported into downstream systems.

2. In the **Properties** view, clear the **Create Fluid Zone** check box.

   By clearing the check box, you are specifying that only the blade geometry, and not the volume around the blade, should be sent to downstream cells. The properties should appear as follows:

Properties of Schematic A2: Blade Design

	A	B	D
1	Property	Value	P
2	General		
3	Component ID	Blade Design	
4	Directory Name	BG	
5	Import Options		
6	Create Hub	☐	
7	Create All Blades	☐	
8	Merge Blade Topology	☑	
9	Blade Loft Direction	Streamwise	
10	Shroud Clearance	None	
11	Create Fluid Zone	☐	
12	Blade Extension (%)	2	

You now have a **BladeGen** system that contains a **Blade Design** cell; the latter is presently in an unfulfilled state, as indicated by the question mark. In the next section, you will fulfill the cell requirements by loading the provided geometry file. In general, you could also fulfill the cell requirements by creating a geometry from scratch.

25.5. Reviewing the Centrifugal Compressor Geometry in BladeGen

This section involves using BladeGen in ANSYS Workbench. BladeGen is a geometry creation tool specifically designed for turbomachinery blades. In this section, you will simply load the geometry from the provided `.bgd` file, and then review the blade design.

1. Double-click the **Blade Design** cell.

   BladeGen opens.

2. In BladeGen, select **File** > **Open** and load the provided file, `Centrifugal_Compressor.bgd`, which is in the working directory.
3. Observe the blade design shown in BladeGen.
4. Optionally exit BladeGen.
5. Return to the **Project Schematic** view.

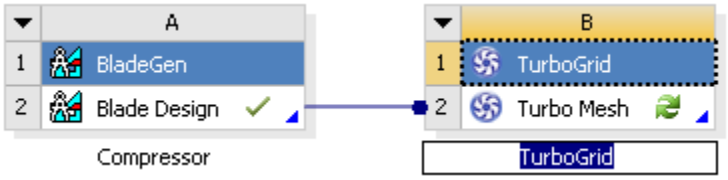
The **Blade Design** cell now displays a green check mark to indicate that the cell is up-to-date. This means that you now have a geometry for the centrifugal compressor that is ready to be used for meshing purposes. In the next section, you will create a CFD-compatible mesh based on this geometry.

25.6. Creating a CFD Mesh using ANSYS TurboGrid

This section involves using ANSYS TurboGrid in ANSYS Workbench. ANSYS TurboGrid is a mesh creation tool specifically designed for turbomachinery blades. In this section, you will use ANSYS TurboGrid to produce a CFD-compatible mesh based on the centrifugal compressor geometry.

1. In the **Project Schematic** view, in the **BladeGen** system, right-click the **Blade Design** cell and select **Transfer Data To New** > **TurboGrid**.

   A **TurboGrid** system opens in the **Project Schematic** view, and is ready to be given a name.



2. Accept the default name by pressing **Enter**.

   The **Turbo Mesh** cell displays a pair of green curved arrows to indicate that the cell has not received the latest upstream data. Normally, you would right-click such a cell and select **Refresh** to transfer data in from the upstream cell. However, for a newly-added Turbo Mesh cell (and whenever ANSYS TurboGrid is not open), this action is not necessary because ANSYS TurboGrid always reads the upstream cell data upon starting up.

   If you were to refresh the **Turbo Mesh** cell, it would show a question mark to indicate that, although the cell's inputs are current, further attention is required in order to bring the cell to an up-to-date status; that further action would typically be to run ANSYS TurboGrid and produce a mesh with it.

3. Double-click the **Turbo Mesh** cell.

   ANSYS TurboGrid opens.

The next several sections guide you through the steps to create a mesh.

25.6.1. Defining the Shroud Tip

For this compressor, the shroud is stationary, and requires a clearance gap between the blade and shroud. Define the tip of the blade using the second blade profile in the blade geometry:

1. In ANSYS TurboGrid, in the object selector, double-click `Geometry` > `Blade Set` > `Shroud Tip`.
2. Apply the following settings:

Tab	Setting	Value
Shroud Tip	Clearance Type > Tip Option	Profile Number
	Tip Profile Number > Tip Profile	2

3. Click **Apply**.

   This defines the shroud tip of the blade (the surface of the blade that is nearest to the shroud).

25.6.2. Creating the Topology

In ANSYS TurboGrid, you may choose a topology pattern based on the type of machine being analyzed. For this geometry, the H/J/C/L-Grid topology will be used.

1. In the object selector, double-click `Topology Set`.
2. Apply the following settings:

Tab	Setting	Value
Definition	Topology Definition > Placement	Traditional with Control Points
	Topology Definition > Method	H/J/C/L-Grid
	Include O-Grid > Width Factor	0.2[a]
	Tip Topology > Shroud	H-Grid Not Matching

[a]A reduced O-Grid width helps to improve mesh quality where the blade thickness is a large fraction of the passage width, as is the case at the upstream end of the hub.

3. Click **Apply** to set the topology.
4. Right-click `Topology Set` and turn off **Suspend Object Updates**.

   After a short time, the topology is generated.
5. Click **Freeze** to freeze the topology settings.

   This prevents unintended automatic changes to the topology.

You now have a mesh topology that requires some adjustments before it is suitable for use in creating a mesh. You can see a 2D preview of the mesh on each of the two layers.

25.6.3. Modifying the Topology

The mesh previews on the hub and shroud tip layers reveal areas that have low mesh angles. You will increase mesh orthogonality in those areas by using control points. You will also adjust the mesh density locally by using edge split controls.

25.6.3.1. Modifying the Hub Layer

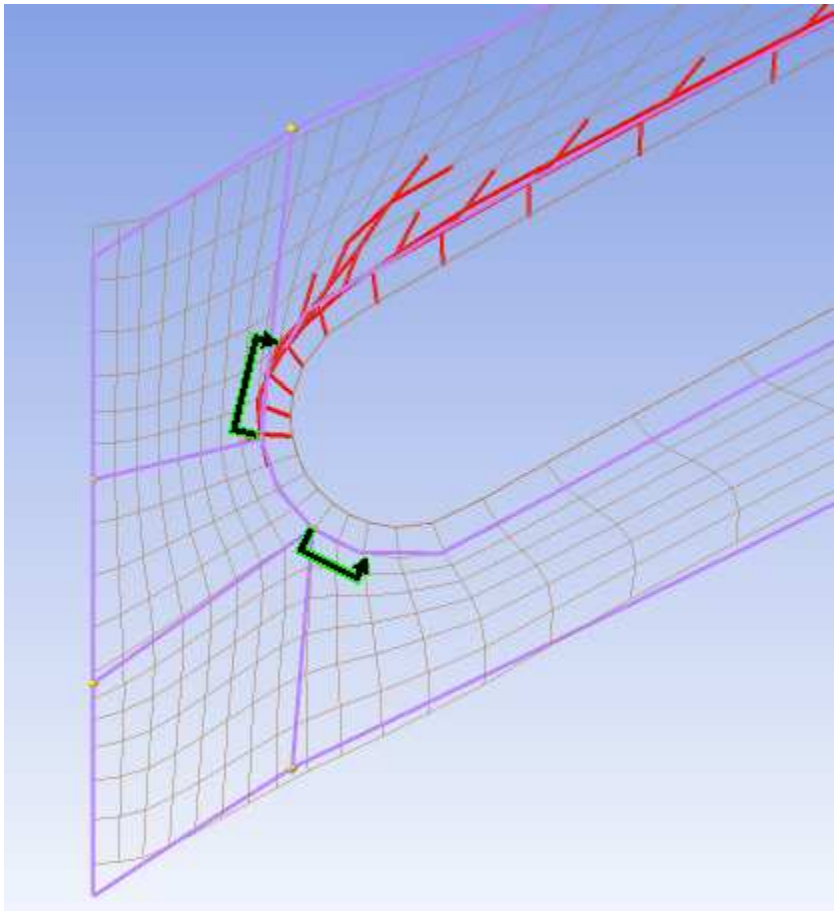
1. Select **Display** > **Blade-to-Blade View** > **Use Passage Excluding Tip Transform**.

   The `Passage Excluding Tip` transform affects how blade-to-blade coordinates are calculated in preparation for viewing with the Blade-to-Blade (Theta-M') transform (in the next step). Compared to the `Full` transform, the `Passage Excluding Tip` transform usually exhibits less distortion in the viewer for blades that have a tip that varies in span. By choosing a transform explicitly, you prevent ANSYS TurboGrid from selecting one of these transforms automatically.
2. Right-click a blank area in the viewer and select **Transformation** > **Blade-to-Blade (Theta-M')**.
3. Click *Hide all geometry objects*  .
4. Turn off the visibility of `Layers` > `Shroud Tip` (by clearing the check box next to it) to make the hub topology more visible.

5. Open `Layers` > `Hub` in the object editor and note that, in the object editor, the mesh measures are shown in red text.
6. Double-click `Minimum Face Angle` to highlight the areas of the mesh that have the smallest angles.

   The areas that have the smallest angles are marked with red lines in the viewer.
7. Zoom in on the leading edge as shown in *Figure 25.1* (p. 450).
8. Click *Select* then move the control points as shown by the displacement vectors in *Figure 25.1* (p. 450).

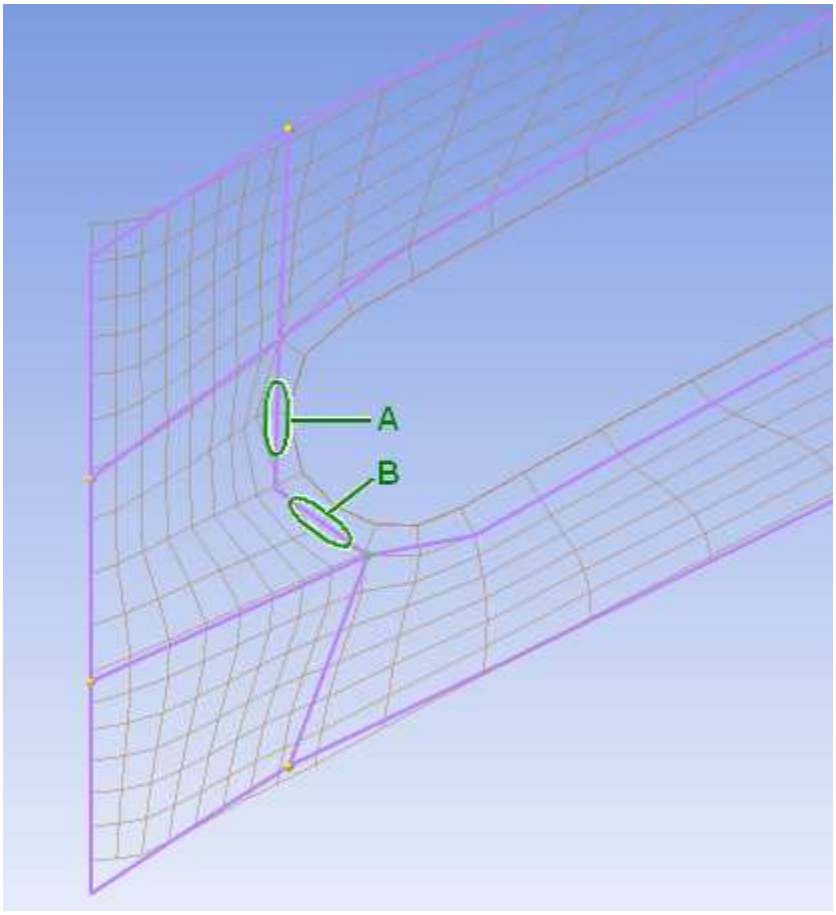
Figure 25.1 Modifying Control Points on the Hub Layer



Confirm that the `Minimum Face Angle` and `Maximum Face Angle` mesh measures have improved for the hub layer.

9. Right-click the master topology line marked "A" in *Figure 25.2* (p. 451) and select **Insert Edge Split Control** from the shortcut menu.

Figure 25.2 Adding Edge Split Controls near the Leading Edge on the Hub Layer



10. In the object editor, change **Split Factor** to `2.0`.
11. Click **Apply**.

    This causes more elements to be placed along the topology line marked "A" in the figure.
12. Using the same technique, add an edge split control with the same split factor at the topology line marked "B" in *Figure 25.2* (p. 451).

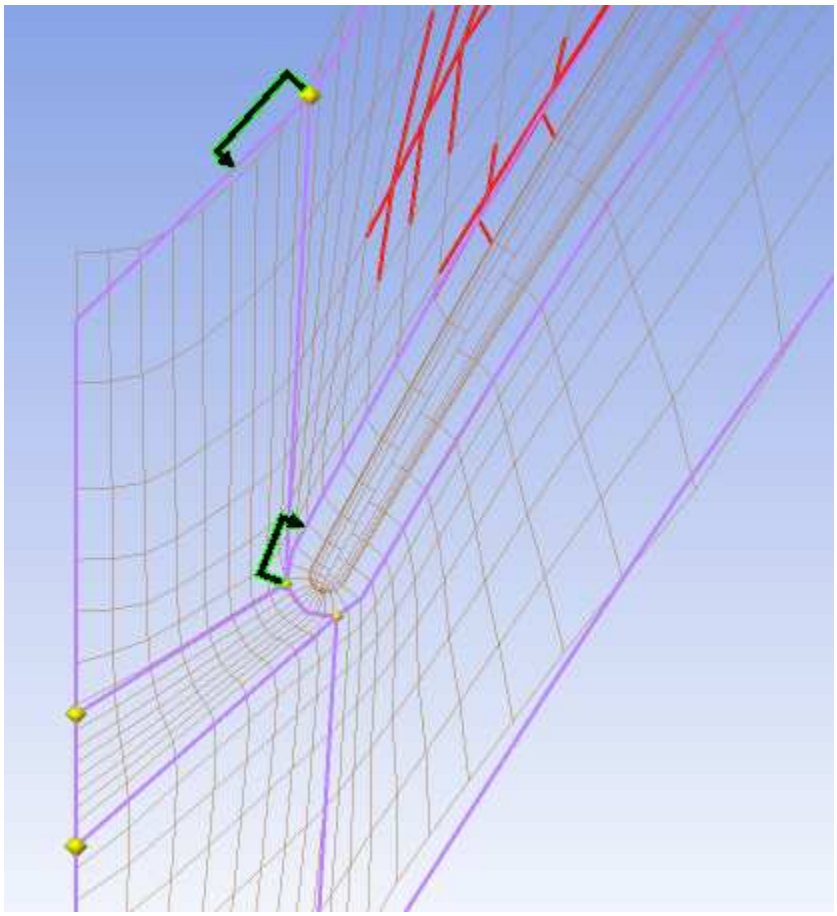
The hub layer is now acceptable for the purposes of this tutorial. Note that it is normal for mesh cells next to the blade to have extremely high aspect ratios. Note also that some elements near the trailing edge appear "crooked" or "wavy"; this is an effect caused by inaccuracies in the viewer transformation to Blade-to-Blade (Theta-M') coordinates, and will not affect the resulting mesh.

25.6.3.2. Modifying the Shroud Tip Layer

1. Turn off the visibility of `Layers` > `Hub`.
2. Turn on the visibility of `Layers` > `Shroud Tip`.
3. Click *Fit View*  .
4. Open `Layers` > `Shroud Tip` in the object editor.
5. Double-click `Minimum Face Angle` to highlight the areas of the mesh that have the smallest angles.
6. Zoom in on the leading edge as shown in *Figure 25.3* (p. 452).

7. Click *Select* then move the control points as shown by the displacement vectors in *Figure 25.3* (p. 452).

Figure 25.3 Modifying Control Points on the Shroud Tip Layer



8. Confirm that the `Minimum Face Angle` and `Maximum Face Angle` mesh measures have improved for the shroud tip layer.

The topology has been improved. In the next section, you will set some parameters that affect the mesh node count and distribution.

25.6.4. Specifying Mesh Data Settings

1. Open `Mesh Data` for editing.
2. Apply the following settings:

Tab	Setting	Value
Mesh Size	Method	Target Passage Mesh Size
	Node Count	Medium (100000)
	Inlet Domain	(Selected)
	Outlet Domain	(Selected)
Passage	Spanwise Blade Distribution Parameters > Method	Element Count and Size

Tab	Setting	Value
	Spanwise Blade Distribution Parameters > # of Elements	25
	Spanwise Blade Distribution Parameters > Const Elements	11
	O-Grid > Method	Element Count and Size
	O-Grid > # of Elements	6
Inlet/Outlet	Inlet Domain > Override default # of Elements	(Selected)
	Inlet Domain > Override default # of Elements > # of Elements	50
	Outlet Domain > Override default # of Elements	(Selected)
	Outlet Domain > Override default # of Elements > # of Elements	25

3. Click **Apply**.

Note

Your mesh quality could decrease slightly after increasing the node count. If so, you might want to make minor adjustments to the hub and shroud control points to improve the quality of your mesh before saving it and using it in the aerodynamic simulation that follows.

4. Open `Layers` for editing.
5. In the list of layers in the object editor, right-click `Hub` and select **Insert Layer After** to insert a layer midway between the hub and shroud tip layers.

   This helps to guide the mesh along the blade in the spanwise direction.

25.6.5. Generating the CFD Mesh

With the topology and mesh data defined, the next step is to create a mesh.

1. Click **Insert** > **Mesh**.

   After a few moments, the mesh is generated.

   You may have noticed that some mesh statistics still show problems. Except for extremely dense meshes, it is normal that the mesh elements next to the walls have very high length ratios and volume ratios. Further actions to improve mesh quality are beyond the scope of this tutorial.
2. Optionally quit ANSYS TurboGrid.
3. Return to the **Project Schematic** view.

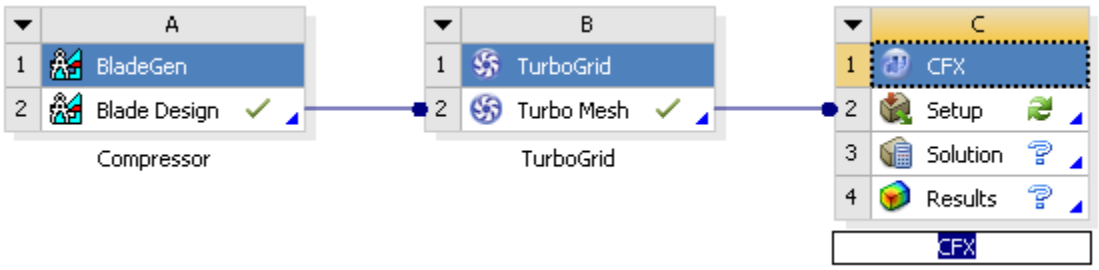
25.7. Defining an Aerodynamic Simulation using CFX-Pre

This section involves using CFX-Pre in ANSYS Workbench. CFX-Pre is a CFD physics preprocessor that has a Turbo Mode facility for setting up turbomachinery CFD simulations. In this section, you will use CFX-Pre in Turbo Mode to define a CFD simulation based on the centrifugal compressor mesh that you created earlier.

25.7.1. Creating a New Case

1. In the **Project Schematic** view, in the **TurboGrid** system, right-click the **Turbo Mesh** cell and select **Transfer Data To New** > **CFX**.

   A **CFX** system opens in the **Project Schematic** view, and is ready to be given a name.



2. Accept the default name by pressing **Enter**.

   The **Setup** cell displays a pair of green curved arrows to indicate that the cell has not received the latest upstream data. Since you have not yet opened CFX-Pre, you can disregard the cell status; CFX-Pre will automatically read the upstream cell data upon starting up *for the first time* (from this cell).

3. Double-click the **Setup** cell.

   CFX-Pre opens.

4. Select **Tools** > **Turbo Mode**.

5. In **Basic Settings** panel, set **Machine Type** to `Centrifugal Compressor`, **Rotation Axis** to Z, and leave the other settings at their defaults.

6. Click **Next**.

7. On the **Component Definition** panel, select **R1** and set the following values:

Setting	Value
Component Type > Type	Rotating
Component Type > Value	22360 [rev min^-1]
Mesh > Available Volumes > Volumes	Inlet, Outlet, Passage Main
Wall Configuration	(Selected)
Wall Configuration > Tip Clearance at Shroud	Yes

   Leave the other settings at their defaults.

8. Click **Next**.

9. On the **Physics Definition** panel, set the following values:

Setting	Value
Fluid	Air Ideal Gas
Analysis Type > Type	Steady State
Model Data > Reference Pressure	1 [atm]
Inflow/Outflow Boundary Templates > P-Total Inlet Mass Flow Outlet	(Selected)
Inflow/Outflow Boundary Templates > Inflow > P-Total	0 [atm]
Inflow/Outflow Boundary Templates > Inflow > T-Total	20 [C]
Inflow/Outflow Boundary Templates > Inflow > Flow Direction	Cylindrical Components
Inflow/Outflow Boundary Templates > Inflow Direction (a, r, t)	1, 0, 0
Inflow/Outflow Boundary Templates > Outflow > Mass Flow	Per Component
Inflow/Outflow Boundary Templates > Outflow > Mass Flow Rate	0.167 [kg s^-1]
Solver Parameters > Convergence Control	Physical Timescale
Solver Parameters > Physical Timescale	0.0002 [s]

Leave the other settings at their defaults.

10. Click **Next**.
11. On the **Interface Definition** panel, verify that each interface is set correctly; click an interface listed in the tree view and then examine the associated settings (shown in the lower portion of the panel) and highlighted regions in the viewer.

    If no regions appear highlighted in the viewer, ensure that highlighting is turned on in the viewer toolbar.
12. Click **Next**.
13. On the **Boundary Definition** panel, verify that each boundary is set correctly; click a boundary listed in the tree view and then examine the associated settings and highlighted regions.
14. Click **Next**.
15. On the **Final Operations** panel, leave the operation set to `Enter General Mode` and click **Finish**.
16. Optionally quit CFX-Pre.
17. Return to the **Project Schematic** view.

25.8. Obtaining a Solution to the Aerodynamic Simulation using CFX-Solver

Generate a solution for the CFD simulation that you just prepared:

1. Right-click the **Solution** cell of the **CFX** system and select **Update**.

After some time, a CFD solution will be generated. If the progress indicator is not visible, you can display it by clicking Show Progress or, to see detailed output, right-click the **Solution** cell and select **Display Monitors**.

2. After the solution has been generated, return to the **Project Schematic** view.

25.9. Viewing the Results of the Aerodynamic Simulation in CFD-Post

CFD-Post enables you to view the results in various ways, including tables, charts, and figures. You can present the results in the form of a report that can be viewed in CFD-Post, or exported for viewing in another application.

Create a report and examine some of the results as follows:

1. Double-click the **Results** cell.

   CFD-Post opens.

2. Click **File** > **Report** > **Load 'Centrifugal Compressor Rotor Report' Template**.
3. In the tree view, under `Report`, double-click `Compressor Performance Results Table`.

   This table presents measures of aerodynamic performance including required power and efficiencies.

4. Double-click **Blade Loading Span 50**.

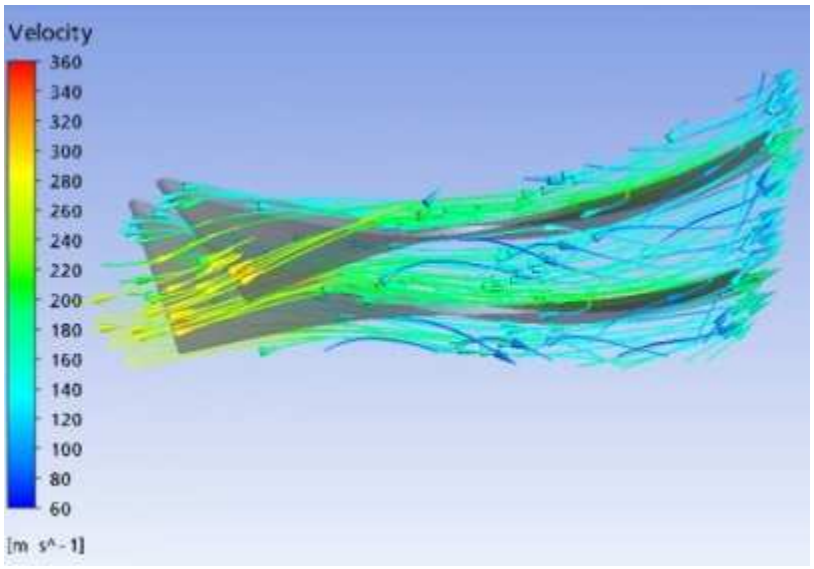
   This is a plot of the pressure vs. streamwise distance along both the pressure and suction sides of the blade at mid-span.

5. Double-click **Streamwise Plot of Pt and P**.

   This is a plot of the streamwise variation of pressure and total pressure.

6. Double-click **Velocity Streamlines Stream Blade TE View**.

   This is a trailing-edge view of the streamlines that start upstream of the blade.



7. To view a full report, click the **Report Viewer** tab found near the bottom right of the window.

A report will be generated (this may take a few minutes) that includes all figures available under `Report` in the tree view. This report can be viewed in CFD-Post or published to be viewed externally as an `.html` or `.txt` file.

Note that if you have visited the **Report Viewer** tab before loading the template, or have otherwise made any changes to the report definition after first viewing the report, you need to click Refresh in the Report Viewer to update the report as displayed.

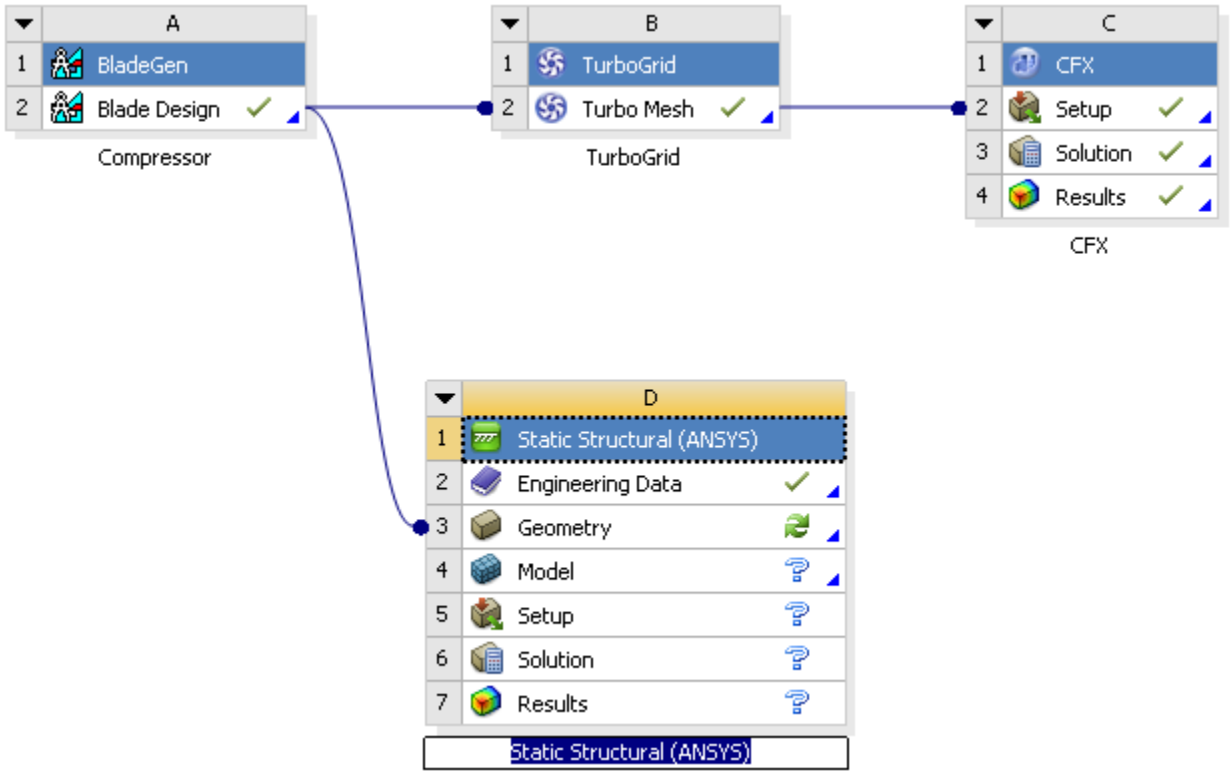
8. Optionally quit CFD-Post.
9. Return to the **Project Schematic** view.

25.10. Simulating Structural Stresses due to Pressure Loads

This section describes the steps required to simulate structural stresses on the blade due to pressure loads from the aerodynamic analysis.

1. In the **Project Schematic** view, in the **BladeGen** system, right-click the **Blade Design** cell and select **Transfer Data To New** > **Static Structural**.

   A **Static Structural** system opens in the **Project Schematic** view, and is ready to be given a name.



2. Accept the default name by pressing **Enter**.
3. Right-click the **Geometry** cell and select **Update**.

   DesignModeler runs in the background, and imports the geometry from BladeGen.

4. Right-click the **Model** cell and select **Update**.

   The Mechanical application runs in the background, and imports the solid model of the centrifugal compressor.

5. Drag the **Solution** cell of the **CFX** system to the **Setup** cell of the **Static Structural** system. Select **Transfer Default Data** to import the pressure data.

   This makes the CFD solution data available for use in a structural analysis.

6. Double-click the **Setup** cell. If a dialog box asks you if you want to read the upstream data, click **Yes**.

   The Mechanical application opens so that you can prepare a static structural analysis.

7. Right-click `Project` > `Model (D4)` > `Static Structural (D5)` > `Imported Load (Solution)` in the tree view and select **Insert** > **Pressure**.

   After a short time, `Project` > `Model (D4)` > `Static Structural (D5)` > `Imported Load (Solution)` > `Imported Pressure` will appear and will be selected.

8. In the details view, click the field beside **Scope** > **Geometry**, then select the blade surface (the large surface; not the thin surface along the edge) by clicking it in the viewer.

9. In the details view next to **Scope** > **Geometry**, click **Apply**.

   The field beside **Geometry** should now display the text "1 Face". You have now chosen the solid model surface onto which to apply the CFD pressure data.

10. In the details view, click the field beside **Transfer Definition** > **CFD Surface**, and select `R1 Blade` from the drop-down.

    You have now chosen the CFD boundary from which to get the CFD pressure data.

11. Right-click `Project` > `Model (D4)` > `Static Structural (D5)` in the tree view and select **Insert** > **Fixed Support**.

12. In the viewer, right-click a blank area of the screen and select **View** > **Bottom**.

13. Select the long thin face of the blade that is at the forefront of the displayed geometry.

14. In the details view next to **Scope** > **Geometry**, click **Apply**.

    The face that you selected is now connected to the hub.

15. In the tree view, right-click `Project` > `Model (D4)` > `Static Structural (D5)` > `Solution (D6)` and select **Insert** > **Stress** > **Equivalent (von-Mises)**.

16. From the tool bar, click *Solve*  and wait for the solver to finish.

17. To verify that the pressure data was applied correctly to the blade, inspect `Project` > `Model (D4)` > `Static Structural (D5)` > `Imported Load (Solution)` > `Imported Pressure` > `Imported Load Transfer Summary`.

    The significant discrepancy shown in the load transfer summary is due to a difference in geometry; the surface onto which the CFX pressures were mapped does not include the blade tip or trailing-edge surfaces. In general, accurate load mapping requires that the surfaces match in terms of geometry and the length scale of mesh elements.

18. Click `Project` > `Model (D4)` > `Static Structural (D5)` > `Solution (D6)` > `Equivalent Stress` to prepare to examine the von-Mises stress results.

19. Animate the physical deformation of the blade along with the associated von-Mises stress results, by clicking *Play*  in the **Graph** window.

Note

You may need to drag the viewer window up in order to see the **Graph** window.

20. Right-click `Project` > `Model (D4)` > `Static Structural (D5)` > `Solution (D6)` and select **Insert** > **Deformation** > **Total**.
21. From the tool bar, click *Solve* Solve.

    When the solver has finished, select `Project` > `Model (D4)` > `Static Structural (D5)` > `Solution (D6)` > `Total Deformation` to prepare to examine the Total Deformation solution.
22. Animate the physical deformation by clicking the **Play** button in the **Graph** window.
23. Select **File** > **Close Mechanical**.
24. Return to the **Project Schematic** view.

25.11. Simulating Structural Stresses due to Rotation

This section shows how to add inertial effects due to rotation.

1. Right-click the upper-left corner of the **Static Structural** system and select **Duplicate**.

   A second **Static Structural** system appears, with its **Geometry**, and **Model** cells taking information from the same sources as in the original system.
2. Rename the newly created system from `Copy of Static Structural` to `With Rotation`.
3. Right-click the **Model** cell in the new system and select **Update**.

   Wait until you see a green check mark in that cell.
4. Double-click the **Setup** cell in the new system.

   The Mechanical application opens.
5. Make sure that `Project` > `Model (E4)` > `Static Structural (E5)` is selected in the tree view, then click the **Inertial** button in the toolbar and select **Rotational Velocity**.
6. From the main menu, select **Units** > **RPM**.
7. In the details view for `Rotational Velocity`, change **Definition** > **Define By** to `Components`.
8. In the details view, set **Definition** > **Z Component** to `22360`.

   (The value will become `22360 RPM (ramped)` when you finish entering the number.)
9. From the tool bar, click *Solve* Solve.
10. To animate the total deformation or equivalent stress of the blade, select the corresponding object (either `Total Deformation` or `Equivalent Stress`, respectively) under `Project` > `Model (E4)` > `Static Structural (E5)` > `Solution (E6)` and click *Play* in the **Graph** window.
11. Select **File** > **Close Mechanical**.
12. Return to the **Project Schematic** view.
13. From the tool bar, click Save to save the project.

14. Exit ANSYS Workbench.

Chapter 26: Axial Turbine Equilibrium and Non-Equilibrium Steam Predictions

This tutorial includes:

26.1. Tutorial Features

In this tutorial you will learn about:

- Selection of material properties from the International Association for the Properties of Water and Steam (IAPWS) database.
- Setting property table ranges.
- Setting up an equilibrium steam calculation.
- Reviewing solution variables particular to the equilibrium solution.
- Setting up a non-equilibrium steam calculation.
- Post-processing features special to the non-equilibrium solution.
- Reviewing solution variables particular to the non-equilibrium solution.

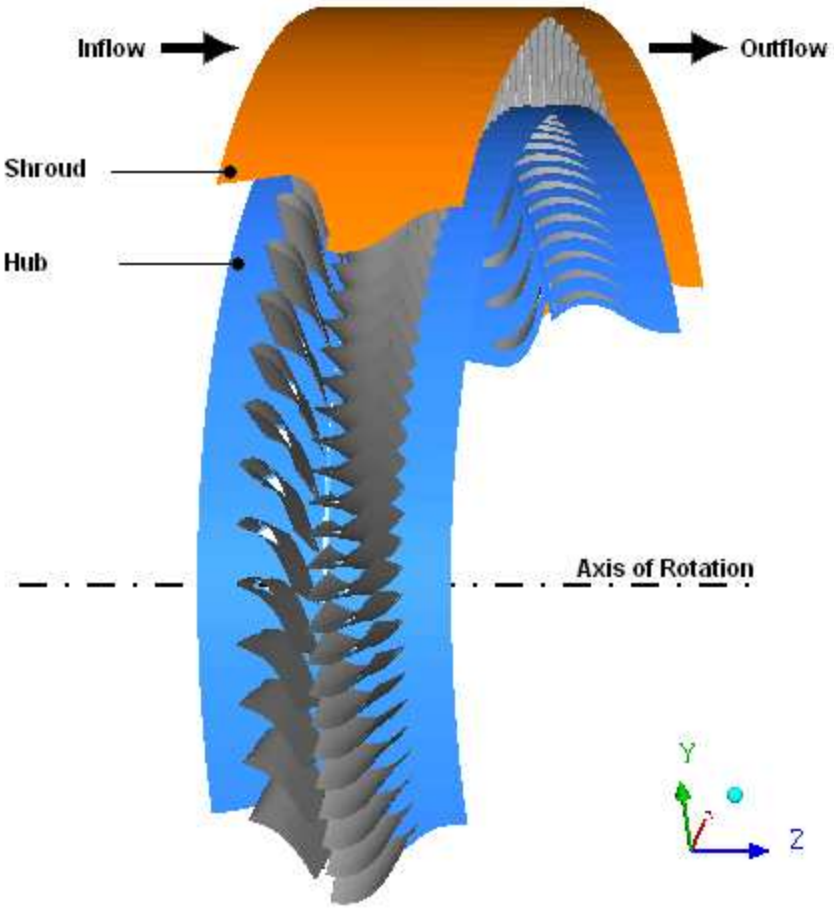
Component	Feature	Details
CFX-Pre	User Mode	Turbo Wizard
	Machine Type	Axial Turbine
	Fluid Type	Binary Homogeneous Mixture (equilibrium solution)
		Multi-Fluid Model (non-equilibrium solution)
	Domain Type	Multiple Domain
	Turbulence Model	k-Epsilon
	Heat Transfer	Total Energy (equilibrium solution)

Component	Feature	Details
		Fluid Dependent (non-equilibrium solution)
	Boundary Conditions	Inlet (subsonic): Total Pressure/Temperature and Mass Fraction(s) (equilibrium solution)
		Inlet (subsonic): Total Pressure/Temperature, Volume Fractions and Droplet Number (non-equilibrium solution)
		Outlet: Static Pressure
	Domain Interfaces	Frozen Rotor
		Periodic
	Timestep	Physical Time Scale
	Material Properties	IAPWS Water Database
	Fluid Pair Models	Small droplet heat transfer, small droplet phase change (non-equilibrium solution)
	Fluid Models	Nucleation, small droplet temperature, droplets with phase change (non-equilibrium solution)
CFD-Post	Location	Turbo Surface
	Plots	Contour

26.2. Overview of the Problem to Solve

The following tutorial uses an axial turbine to demonstrate setting up and executing equilibrium and non-equilibrium steam calculations using the IAPWS water database for properties.

The full geometry contains 60 stator blades and 113 rotor blades. The following figure shows approximately half of the full geometry. The inflow average velocity is in the order of 100 m/s.

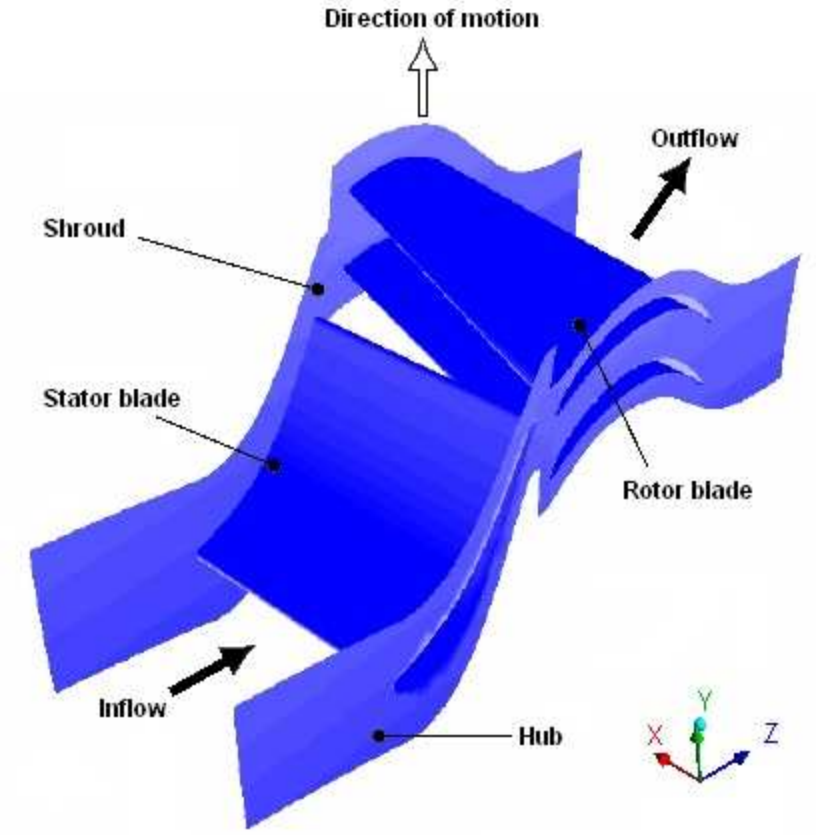


The geometry to be modeled consists of a single stator blade passage and two rotor blade passages. This is an approximation to the full geometry since the ratio of rotor blades to stator blades is close to, but not exactly, 2:1. In the stator blade passage a 6° section is being modeled (360°/60 blades), while in the rotor blade passage, a 6.372° section is being modeled (2*360°/113 blades). This produces a pitch ratio at the interface between the stator and rotor of 0.942. As the flow crosses the interface, it is scaled to allow this type of geometry to be modeled. This results in an approximation of the inflow to the rotor passage. Furthermore, the flow across the interface will not appear continuous due to the scaling applied.

In this example, the rotor rotates at 523.6 rad/s about the Z-axis while the stator is stationary. Periodic boundaries are used to allow only a small section of the full geometry to be modeled.

The important problem parameters are:

- Total inlet pressure = 0.265 bar
- Static outlet pressure = 0.0662 bar
- Total inlet temperature = 328.5 K



In this tutorial, you will generate two steady-state solutions: one using a multicomponent fluid consisting of a homogeneous binary mixture of liquid water and water vapor, the other using two separate phases to represent liquid water and water vapor in a non-equilibrium simulation. The solution variables particular to the equilibrium and non-equilibrium solutions will be processed in order to understand the differences between the two solutions.

26.3. Before You Begin

If this is the first tutorial you are working with, it is important to review the following topics before beginning:

- *Preparing a Working Directory and Starting ANSYS CFX in Standalone Mode* (p. 1)
- *Running ANSYS CFX Tutorials in ANSYS Workbench* (p. 2)
- *Changing the Display Colors* (p. 5)
- *Playing a Tutorial Session File* (p. 4)

26.4. Starting CFX-Pre

1. Prepare the working directory using the following files in the `examples` directory:
   - `WaterVaporEq.pre`
   - `WaterVaporEq.cfx`
   - `WaterVaporNonEq.pre`
   - `stator.gtm`

- `rotor.grd`

2. Set the working directory and start CFX-Pre.

   For details, see *Preparing a Working Directory and Starting ANSYS CFX in Standalone Mode* (p. 1)

26.5. Defining the Equilibrium Case in CFX-Pre

The following sections describe the equilibrium simulation setup in CFX-Pre.

If you want to set up the simulation automatically and continue to *Obtaining a Solution for the Equilibrium Case* (p. 471), run `WaterVaporEq.pre`.

This tutorial uses the Turbomachinery wizard in CFX-Pre. This pre-processing mode is designed to simplify the setup of turbomachinery simulations.

1. In CFX-Pre, select **File** > **New Case**.
2. Select **Turbomachinery** and click **OK**.
3. Select **File** > **Save Case As**.
4. Under **File name**, type `WaterVaporEq`.
5. If you are notified the file already exists, click **Overwrite**.

   This file is provided in the tutorial directory and may exist in your working directory if you have copied it there.
6. Click **Save**.

26.5.1. Basic Settings

1. Set **Machine Type** to `Axial Turbine`.
2. Click **Next**.

26.5.2. Component Definition

Two new components are required. As you specify them, CFX-Pre imports the meshes.

1. Right-click a blank area near the **Component Definition** tree and select **Add Component** from the shortcut menu.
2. Create a new component of type `Stationary`, named `S1`.
3. Apply the following setting:

Setting	Value
Mesh > File	stator.gtm [1 (p. 465)]
1. You may have to select the `CFX Mesh (*gtm *cfx)` option under **Files of type** in order to see the file.	

4. Create a new component of type `Rotating`, named `R1`.
5. Set **Component Type** > **Value** to 523.6 [radian s^-1].

6. Click *Browse*  beside **Mesh** > **File**.
7. In the **Import Mesh** dialog box, apply the following settings:

Setting	Value
File name	rotor.grd [1 (p. 466)]
Options > Mesh Units	m

1. You may have to select the `CFX-TASCflow (*grd)` option under **Files of type** in order to see the file.

Note

The components must be ordered as above (stator then rotor) in order for the interface to be created correctly. The order of the two components can be changed, if necessary, by right-clicking `S1` and selecting **Move Component Up**.

When a component is defined, Turbo Mode will automatically select a list of regions that have been recognized as potential boundaries and interfaces. This information should be reviewed in the **Region Information** section, situated below the Mesh section, to ensure that all is correct. This information will be used to help set up boundary conditions and interfaces. The upper case turbo regions that are selected (for example, HUB) correspond to the region names in the CFX-TASCflow grd file. CFX-TASCflow turbomachinery meshes use these names consistently.

8. Click **Open** on the **Import Mesh** dialog box.
9. Click **Next**.

26.5.3. Physics Definition

In this section, you will set properties of the fluid domain and some solver parameters. Note that initially you will choose the fluid to be `Water Ideal Gas`, but later you will create a new fluid based on the IAPWS database for water and override this initial setting with it.

1. Apply the following settings

Tab	Setting	Value
Physics Definition	Fluid	Water Ideal Gas
	Analysis Type > Type	Steady State
	Model Data > Reference Pressure	0 [atm] [1 (p. 467)]
	Model Data > Heat Transfer	Total Energy
	Model Data > Turbulence	k-Epsilon
	Inflow/Outflow Boundary Templates > P-Total Inlet P-Static Outlet	(Selected)
	Inflow/Outflow Boundary Templates > Inflow > P-Total	0.265 [bar]

Tab	Setting	Value
	Inflow/Outflow Boundary Templates > Inflow > T-Total	328.5 [K] [2 (p. 467)]
	Inflow/Outflow Boundary Templates > Inflow > Flow Direction	Normal to Boundary
	Inflow/Outflow Boundary Templates > Outflow > P-Static	0.0662 [bar]
	Interface > Default Type	Frozen Rotor
	Solver Parameters > Convergence Control	Physical Timescale
	Solver Parameters > Physical Timescale	0.0005 [s] [3 (p. 467)]

1. Because this tutorial involves vaporization, you should use absolute pressures throughout. This can be accomplished by setting the reference pressure to 0 atm.
2. From the problem description.
3. The physical timescale that will be set up is derived from the rotational speed of the 113 rotor blades. See the CFX Best Practices Guide for Turbomachinery in the *CFX Reference Guide* for an explanation of how this value is calculated.

2. Click **Next**.

26.5.4. Interface Definition

CFX-Pre will try to create appropriate interfaces using the region names viewed previously in the **Region Information** section (in the **Component Definition** setup screen). In this case, you should see that a periodic interface has been generated for both the rotor and the stator. The generated periodic interface can be edited or deleted. Interfaces are required when modeling a small section of the true geometry. An interface is also needed to connect the two components together across the frame change.

1. Review the various interfaces but do not change them.
2. Click **Next**.

26.5.5. Boundary Definition

CFX-Pre will try to create appropriate boundary conditions using the region names presented previously in the **Region Information** section. In this case, you should see a list of generated boundary conditions. They can be edited or deleted in the same way as the interface connections that were set up earlier.

1. Review the various boundary definitions but do not change them.
2. Click **Next**.

26.5.6. Final Operations

1. Set **Operation** to `Enter General Mode`.
2. Click **Finish**.

   After you click **Finish**, a dialog box appears stating that a Turbo report will not be included in the solver file because you are entering General Mode.

3. Click **Yes** to continue.

26.5.7. Defining the Properties of Water

Earlier in the physics definition portion of the Turbomachinery wizard, you specified `Water Ideal Gas` as the fluid in the domain. Here, you will specify a homogeneous binary mixture to replace it. To create the mixture, you will take two pure fluids from the IAPWS database for water and combine them. The pure fluids that will be combined are `H2Og`, representing water vapor, and `H2Ol`, representing liquid water. The mixture will be named `H2Olg`.

The present simulations use the published IAPWS-IF97 (International Association for the Properties of Water and Steam - Industrial Formulation 1997) water tables for properties. The published IAPWS-IF97 equations have been implemented in ANSYS CFX, allowing you to directly select them for use in your simulations. The present example uses these properties in a tabular format requiring you to specify the range of the properties (such as min/max pressure and temperature bounds) and the number of data points in each table. Note that the IAPWS-IF97 properties have been tested for extrapolation into metastable regions, a fact that will be used for the non-equilibrium calculations that require this kind of state information.

1. Click *Material* 
2. Name the new material `H2Og`.
3. Enter the following settings for `H2Og`:

Tab	Setting	Value
Basic Settings	Material Group	IAPWS IF97
	Thermodynamic State	(Selected)
	Thermodynamic State > Thermodynamic State	Gas
Material Properties	Thermodynamic Properties > Table Generation	(Selected)
	Thermodynamic Properties > Table Generation > Minimum Temperature	(Selected)
	Thermodynamic Properties > Table Generation > Minimum Temperature > Min. Temperature	250 [K] [1 (p. 469)]
	Thermodynamic Properties > Table Generation > Maximum Temperature	(Selected)
	Thermodynamic Properties > Table Generation > Maximum Temperature > Max. Temperature	400 [K]
	Thermodynamic Properties > Table Generation > Minimum Absolute Pressure	(Selected)
	Thermodynamic Properties > Table Generation > Minimum Absolute Pressure > Min. Absolute Pres.	0.01 [bar]
	Thermodynamic Properties > Table Generation > Maximum Absolute Pressure	(Selected)
	Thermodynamic Properties > Table Generation > Maximum Absolute Pressure > Max. Absolute Pres.	0.6 [bar]
	Thermodynamic Properties > Table Generation > Maximum Points	(Selected)
	Thermodynamic Properties > Table Generation > Maximum Points > Maximum Points	100

Tab	Setting	Value
1. The H2Og minimum temperature is set to 250 K as the vapor might possibly supercool (in the NES calculations) to temperatures lower than the triple point temperature.		

4. Click **OK**.
5. In the **Outline** tree, under `Materials`, right-click `H2Og` and select **Duplicate**.
6. Rename `Copy of H2Og` to **H2Ol** (using the letter "l" as in "liquid").
7. Open `H2Ol` for editing.
8. On the **Basic Settings** tab, change **Thermodynamic State** > **Thermodynamic State** from `Gas` to `Liquid`.
9. Click **OK**
10. Create a new material named `H2Olg`.
11. Enter the following settings for `H2Olg`:

Tab	Setting	Value
Basic Settings	Options	Homogeneous Binary Mixture
	Material Group	IAPWS IF97
	Material 1	H2Og
	Material 2	H2Ol
Saturation Properties	Option	IAPWS Library
	Table Generation	(Selected)
	Table Generation > Minimum Temperature	(Selected)
	Table Generation > Minimum Temperature > Min. Temperature	273.15 [K] [1 (p. 469)]
	Table Generation > Maximum Temperature	(Selected)
	Table Generation > Maximum Temperature > Max. Temperature	400 [K]
	Table Generation > Minimum Absolute Pressure	(Selected)
	Table Generation > Minimum Absolute Pressure > Min. Absolute Pres.	0.01 [bar]
	Table Generation > Maximum Absolute Pressure	(Selected)
	Table Generation > Maximum Absolute Pressure > Max. Absolute Pres.	0.6 [bar]
	Table Generation > Maximum Points	(Selected)
	Thermodynamic Properties > Table Generation > Maximum Points > Maximum Points	100
1. The minimum temperature is set to 273.15 K due to the fact that the saturation properties implied by H2Ogl are not likely to be used below the triple point temperature.		

12. Click **OK**.

26.5.8. Modifications to Domain and Boundary Conditions

You now need to update the initial setting for the domain fluid (initially set while in the Turbomachinery wizard) with the new homogeneous binary mixture fluid you have just created. This mixture (H2Olg) acts as a container fluid identifying two child materials, H2Ol and H2Og, each representing the liquid and vapor properties in the pure fluid system. The equilibrium solution uses the binary mixture fluid, H2Olg, and assumes that equilibrium conditions relate H2Ol and H2Og at all times. In the non-equilibrium solution (in the second part of this tutorial), H2Ol and H2Og are each used separately to define the fluids that are active in the domain; this is a requirement since the equilibrium constraint is no longer applicable in that case.

1. Open domain R1 for editing.
2. On the **Basic Settings** tab under **Fluid and Particle Definitions**, delete any existing items by selecting them and clicking *Remove selected item*  .
3. Click *Add new item*  .
4. Set **Name** to `H2Olg` and click **OK**.
5. Set **Fluid and Particle Definitions** > **H2Olg** > **Material** to `H2Olg` and click **Apply**.
6. On the **Fluid Models** tab set **Component Models** > **Component** > **H2Og** > **Option** to `Equilibrium Fraction` and click **OK**.
7. Open `Simulation` > `Flow Analysis 1` > `S1` > `S1 Inlet` for editing.
8. On the **Boundary Details** tab, set **Component Details** > **H2Og** > **Mass Fraction** to `1.0`.
9. Click **OK**.

26.5.9. Setting Initial Values

1. Click *Global Initialization*  .
2. Enter the following settings:

Tab	Setting	Value
Global Settings	Initial Conditions > Cartesian Velocity Components > Option	Automatic with Value
	Initial Conditions > Cartesian Velocity Components > U	0 [m s^-1]
	Initial Conditions > Cartesian Velocity Components > V	0 [m s^-1]
	Initial Conditions > Cartesian Velocity Components > W	100 [m s^-1] [1 (p. 471)]
	Initial Conditions > Static Pressure > Option	Automatic with Value
	Initial Conditions > Static Pressure > Relative Pressure	0.2 [bar]
	Initial Conditions > Temperature > Option	Automatic with Value
	Initial Conditions > Temperature > Temperature	328.5 [K] [1 (p. 471)]
	Initial Conditions > Component Details > H2Og > Option	Automatic with Value

Tab	Setting	Value
	Initial Conditions > Component Details > H2Og > Mass Fraction	1.0
1. From the problem description.		

3. Click **OK**.

26.5.10. Writing the CFX-Solver Input (.def) File

1. Click *Define Run*  .
2. Apply the following settings:

Setting	Value
File name	WaterVaporEq.def

3. Click **Save**.

   CFX-Solver Manager automatically starts and, on the **Define Run** dialog box, the **Solver Input File** is set.
4. Quit CFX-Pre, saving the simulation (`.cfx`) file.

26.6. Obtaining a Solution for the Equilibrium Case

When CFX-Pre has shut down, and CFX-Solver Manager has started, obtain a solution to the CFD problem by following the instructions below.

1. In CFX-Solver Manager, click **Start Run**.
2. At the end of the run, on the completion message that appears, select **Post-Process Results**.
3. If using Standalone mode, select **Shut down CFX-Solver Manager**.
4. Click **OK**.

26.7. Viewing the Equilibrium Case Results in CFD-Post

The equilibrium case produces solution variables unique to these model settings. The most important ones are static pressure, mass fraction and temperature, which will be briefly described here.

1. When CFD-Post starts, the **Domain Selector** dialog box might appear. If it does, ensure that both the R1 and S1 domains are selected, then click **OK** to load the results from these domains.
2. Click the **Turbo** tab.
3. The **Turbo Initialization** dialog box is displayed, and asks you whether you want to auto-initialize all components. Click **Yes**.

   The **Turbo** tree view shows the two components in domains R1 and S1. In this case, the initialization works without problems. If there were any problems initializing a component, this would be indicated in the tree view.

Note

If you do not see the **Turbo Initialization** dialog box, or as an alternative to using that dialog box, you can initialize all components by clicking the **Initialize All Components** button which is visible initially by default, or after double-clicking the **Initialization** object in the **Turbo** tree view.

26.7.1. Displaying the Equilibrium Solution Variables using Contour Plots

Make a 2D surface at 50% span to be used as a locator for plots:

1. On the main menu select **Insert** > **Location** > **Turbo Surface** and name it `Turbo Surface 1`.
2. On the **Geometry** tab, set **Method** to `Constant Span` and **Value** to `0.5`.
3. Click **Apply**.
4. Turn off the visibility of `Turbo Surface 1`.

26.7.1.1. Static Pressure and Mass Fraction Contour Plots

In the equilibrium solution, phase transition occurs the moment saturation conditions are reached in the flow. The amount of moisture ultimately created from phase transition is determined in the following manner.

The static pressure solution field yields the saturation enthalpy through the function $h_{\mathrm{sat}}(P)$, available from the IAPWS database. During the solution, if the predicted mixture static enthalpy, h, falls below $h_{\mathrm{sat}}(P)$ at any point, the mass fraction of the condensed phase (also called *wetness*) can be directly calculated also based on the IAPWS properties. The degree to which wetness is generated depends on the amount by which h is less than $h_{\mathrm{sat}}(P)$.

Note that the mass fraction is predicted at each iteration in the solution, and is used to update other two-phase mixture properties required at each step in the solution. The final results therefore include all of the two-phase influences, but assuming equilibrium conditions. The equilibrium solution mass fraction contour plot will be compared to the non-equilibrium solution one in *Viewing the Non-Equilibrium Case Results in CFD-Post* (p. 477).

In the next step, you are asked to create the pressure and mass fraction contour plots.

1. Create a static pressure contour plot on `Turbo Surface 1`:
   a. Create a new contour plot named `Static Pressure`.
   b. In the **Details** view on the **Geometry** tab, set **Locations** to `Turbo Surface 1` and **Variable** to `Pressure`, then click **Apply**.
2. Turn off the visibility of `Static Pressure` when you have finished observing the results.
3. Create a contour plot on `Turbo Surface 1` that shows the mass fraction of the liquid phase:
   a. Create a new contour plot named `Mass Fraction of Liquid Phase`.
   b. In the **Details** view on the **Geometry** tab, set **Locations** to `Turbo Surface 1` and **Variable** to `H2Ol.Mass Fraction`, then click **Apply**.
4. Turn off the visibility of **Mass Fraction of Liquid Phase** when you have finished observing the results.

26.7.1.2. Static Temperature Contour Plots

In the equilibrium solution, the condensed and gas phases share the same temperature and, as a result, predictions of thermodynamic losses are not possible.

In the following step you are asked to view the temperature field in the solution, and note that, due to the equilibrium constraint, it represents conditions for the mixture.

1. Create a contour plot on `Turbo Surface 1` that shows the static temperature.
   a. Create a new contour plot named `Static Temperature`.
   b. In the **Details** view on the **Geometry** tab, set **Locations** to `Turbo Surface 1` and **Variable** to `Temperature`, then click **Apply**.
2. Once you have observed the results save the state and exit CFD-Post.

26.8. Defining the Non-Equilibrium Case in CFX-Pre

The non-equilibrium calculation introduces a number of additional transport equations to the equilibrium solution, namely volume fractions for each phase and droplet number for all condensing phases. In addition, energy equations need to be specified for each of the phases in the solution. The setup of these equations is automated based on model selections to be described subsequently. Important to the predictions are interphase heat and mass transfer between the vapor and condensed phases due to small droplets created by homogeneous nucleation. Selection of the required phase pair conditions is made easier by provision of special small droplet models (where small droplet implies droplet sizes generally below one μm). Phase transition is initiated based on predicted metastable state (measured by supercooling level) conditions in the flow in conjunction with a classical homogeneous nucleation model.

The non-equilibrium solution is therefore closely dependent on the evolving conditions along the flow path leading to phase transition and subsequent strong interaction (by heat/mass transfer) between phases.

If you want to set up the non-equilibrium simulation automatically and continue to *Obtaining a Solution for the Non-Equilibrium Case* (p. 476), run `WaterVaporNonEq.pre`.

26.8.1. Opening the Existing Simulation

1. If CFX-Pre is not already running, start it.
2. Select **File** > **Open Case**.
3. If required, set the path location to the tutorial directory.
4. Select the simulation file `WaterVaporEq.cfx`.
5. Click **Open**.
6. Select **File** > **Save Case As**.
7. Change the name to `WaterVaporNonEq.cfx`.
8. Click **Save**.

26.8.2. Modifying the Domains

The non-equilibrium case is different from the equilibrium case in that the creation of the second phase (that is, the liquid water) is based on vapor supercooling in conjunction with fluid expansion rate and a

nucleation model. The location where the phase transition happens is not specified, but evolves as part of the solution.

In this section, you will specify a nucleation model for the phase that is considered condensable. For details on the nucleation model, see Droplet Condensation Model in the *CFX-Solver Modeling Guide*.

You will also set the droplet number and volume fraction of the condensable phase to zero at the inlet. The condensable phase will appear within the domain by homogeneous nucleation. If this case were to involve wetness at the inlet, you would have a choice of specifying either droplet number or droplet diameter as a boundary along with the volume fraction.

1. Open domain R1.
2. On the **Basic Settings** tab under **Fluid and Particle Definitions**, delete any existing items by selecting them and clicking *Remove selected item*  .
3. Create two new materials named H2Og and H2Ol by using the *Add new item*  icon.
4. Apply the following settings to domain R1:

Tab	Setting	Value
Basic Settings	Fluid and Particle Definitions	H2Og
	Fluid and Particle Definitions > H2Og > Material	H2Og
	Fluid and Particle Definitions > H2Og > Morphology > Option	Continuous Fluid
	Fluid and Particle Definitions	H2Ol
	Fluid and Particle Definitions > H2Ol > Material	H2Ol
	Fluid and Particle Definitions > H2Ol > Morphology > Option	Droplets (Phase Change)
Fluid Models	Multiphase > Homogeneous Model	(Selected)
	Heat Transfer > Homogeneous Model	(Cleared)
	Heat Transfer > Option	Fluid Dependent
Fluid Specific Models	Fluid	H2Og
	Fluid > H2Og > Heat Transfer Model > Option	Total Energy
	Fluid	H2Ol
	Fluid > H2Ol > Heat Transfer Model > Option	Small Droplet Temperature
	Fluid > H2Ol > Nucleation Model	(Selected)
	Fluid > H2Ol > Nucleation Model > Option	Homogeneous
	Fluid > H2Ol > Nucleation Model > Nucleation Bulk Tension Factor [1 (p. 475)]	(Selected)
	Fluid > H2Ol > Nucleation Model > Nucleation Bulk Tension Factor > Nucleation Bulk Tension	1.0
Fluid Pair Models	Fluid Pair	H2Og | H2Ol
	Fluid Pair > H2Og | H2Ol > Interphase Transfer > Option	Particle Model
	Fluid Pair > H2Og | H2Ol > Mass Transfer > Option	Phase Change

Tab	Setting	Value
	Fluid Pair > H2Og | H2Ol > Mass Transfer > Phase Change Model > Option	Small Droplets
	Fluid Pair > H2Og | H2Ol > Heat Transfer > Option	Small Droplets

1. The **Nucleation Bulk Tension Factor** scales the bulk surface tension values used in the nucleation model. Classical nucleation models are very sensitive to the bulk surface tension, and only slight adjustments will modify the nucleation rate quite significantly. It is common practice in CFD simulations to alter the bulk surface tension values slightly in order to bring results in-line with experiment. Studies suggest that with the IAPWS database and conditions less than 1 bar, a **Nucleation Bulk Tension Factor** of 1.0 is the best first setting.

Note

The small droplet setting for H2Ol heat transfer implies that the temperature is algebraically determined as a function of the droplet diameter, which in turn is calculated from other solution variables such as H2Ol volume fraction and droplet number.

5. Click **OK**.
6. Apply the same settings to domain S1.

   Most of the settings will have been already copied from domain R1 to domain S1, however the nucleation model settings must be set explicitly.
7. Open S1 Inlet and enter the following settings:

Tab	Setting	Value
Boundary Details	Heat Transfer > Option	Fluid Dependent
Fluid Values	Boundary Conditions	H2Og
	Boundary Conditions > H2Og > Heat Transfer > Option	Total Temperature
	Boundary Conditions > H2Og > Heat Transfer > Total Temperature	328.5 [K]
	Boundary Conditions > H2Og > Volume Fraction > Option	Value
	Boundary Conditions > H2Og > Volume Fraction > Volume Fraction	1.0
	Boundary Conditions	H2Ol
	Boundary Conditions > H2Ol > Volume Fraction > Option	Value
	Boundary Conditions > H2Ol > Volume Fraction > Volume Fraction	0
	Boundary Conditions > H2Ol > Droplet Number > Option	Specified Number [1 (p. 475)]
	Boundary Conditions > H2Ol > Droplet Number > Droplet Number	0 [m^-3]

1. Not Specified Diameter.

8. Click **OK**.
9. Click *Solver Control*  .
10. On the **Basic Settings** tab, set **Convergence Control** > **Fluid Timescale Control** > **Physical Timescale** to `5e-005 [s]` and click **OK**.

    Because the non-equilibrium simulation involves vapor and therefore tends to be unstable, it is recommended that you set the physical timescale to a relatively small value. The value set here was found to be suitable for this simulation by trial and error.

26.8.3. Writing the CFX-Solver Input (.def) File

1. Click *Define Run*  .
2. Apply the following settings:

Setting	Value
File name	WaterVaporNonEq.def

3. Click **Save**.

   CFX-Solver Manager automatically starts and, on the **Define Run** dialog box, the **Solver Input File** is set.
4. Quit CFX-Pre, saving the simulation (`.cfx`) file at your discretion.

26.9. Obtaining a Solution for the Non-Equilibrium Case

The **Define Run** dialog box will be displayed when CFX-Solver Manager launches. **CFX-Solver Input File** will already be set to the name of the CFX-Solver input file just written.

1. Click **Start Run**.

   CFX-Solver runs and attempts to obtain a solution. This can take some time depending on your system. Eventually a dialog box appears.

   Note

   You may notice messages in the solver output regarding problems with evaluating material properties. This is a result of the absolute pressure reaching values outside the range of internal material property tables. In this case, the messages are temporary and stop appearing well before convergence. If you encounter a simulation where the messages persist, or you otherwise suspect that the results might be adversely affected, you can change the ranges of internal material property tables by editing the relevant materials in CFX-Pre.

2. At the end of the run, on the completion message that appears, select **Post-Process Results**.
3. If using Standalone mode, select **Shut down CFX-Solver Manager**.
4. Click **OK**.

26.10. Viewing the Non-Equilibrium Case Results in CFD-Post

The non-equilibrium calculation creates a number of solution variables. The most important ones, which are briefly described in the following section, are supercooling, nucleation rate, droplet number, mass fraction, and particle diameter.

1. When CFD-Post starts, the **Domain Selector** dialog box might appear. If it does, ensure that both the R1 and S1 domains are selected, then click **OK** to load the results from these domains.
2. Click the **Turbo** tab.
3. The **Turbo Initialization** dialog box is displayed, and asks you whether you want to auto-initialize all components. Click **Yes**.

   The **Turbo** tree view shows the two components in domains R1 and S1. In this case, the initialization works without problems. If there were any problems initializing a component, this would be indicated in the tree view.

Note

If you do not see the **Turbo Initialization** dialog box, or as an alternative to using that dialog box, you can initialize all components by clicking the **Initialize All Components** button which is visible initially by default, or after double-clicking the **Initialization** object in the **Turbo** tree view.

26.10.1. Displaying the Non-Equilibrium Solution Variables using Contour Plots

Make a 2D surface at 50% span to be used as a locator for plots:

1. On the main menu select **Insert** > **Location** > **Turbo Surface** and name it `Turbo Surface 1`.
2. On the **Geometry** tab, set **Method** to `Constant Span` and **Value** to `0.5`.
3. Click **Apply**.
4. Turn off the visibility of `Turbo Surface 1`.

26.10.1.1. Supercooling Contour Plot

The non-equilibrium solution provides considerable detail on conditions related to phase transition. In particular, the solution tracks the evolution of metastable conditions in the flow through a supercooling variable obtained on the basis of local pressure and gas phase temperature. The supercooling level is the primary variable influencing the nucleation model. The nucleation model provides an estimate of the rate of production of critically sized nuclei that are stable enough to grow in a supercooled vapor flow.

In the next step, you will plot the degree of supercooling, where the supercooling represents the difference between the saturation temperature, set by the local pressure in the flow, and the associated local gas phase temperature. At the inlet, supercooling is often shown as negative, indicating superheated conditions. At equilibrium conditions, no supercooling is allowed since liquid and vapor phases always share the same temperature. Also, critical homogeneous nucleation at pressures below one atmosphere generally involves supercooling levels of up to 35 to 40 K, depending on rate of expansion in the flow.

1. Create a supercooling contour plot on `Turbo Surface 1`.

This will display the degree of non-equilibrium conditions in the gas phase prior to homogeneous phase transition.

1. Create a new contour plot named `Degree of Supercooling`.
2. In the **Details** view on the **Geometry** tab, set **Locations** to `Turbo Surface 1` and **Variable** to `H2Og.Supercooling` then click **Apply**.

2. Turn off the visibility of `Degree of Supercooling` when you have finished observing the results.

26.10.1.2. Nucleation Rate and Droplet Number Contour Plots

In the next step, you will plot the nucleation rate to show the level of nucleation attained, along with the droplet number concentration that is present in the flow following nucleation. Note that the nucleation rates reach very high levels, with peak values remaining in the flow for only a short time (that is, as long as supercooled conditions remain). The supercooled droplets released at nucleation grow rapidly, taking mass from the vapor phase and releasing thermal energy, which acts to rapidly reduce vapor supercooling to near zero. This fact removes further significant nucleation and is why the phase transition process is limited to a narrow region in the flow in most cases.

1. Create a nucleation rate contour plot on `Turbo Surface 1`.

   This will display the nucleation front at the point of maximum supercooling.

   1. Create a new contour plot named `Nucleation Rate`.
   2. In the **Details** view on the **Geometry** tab, set **Locations** to `Turbo Surface 1` and **Variable** to `H2Ol.Nucleation Rate`.
   3. Set **Range** to `Local` and then click **Apply**.

2. Turn off the visibility of `Nucleation Rate` when you have finished observing the results.
3. Create a droplet number contour plot on `Turbo Surface 1`.

   This will display the predicted droplet concentration resulting from phase transition.

   1. Create a new contour plot named `Droplet Number`.
   2. In the **Details** view on the **Geometry** tab, set **Locations** to `Turbo Surface 1` and **Variable** to `H2Ol.Droplet Number`.
   3. Click **Apply**.

4. Turn off the visibility of **Droplet Number** when you have finished observing the results.

26.10.1.3. Mass Fraction and Particle Diameter Contour Plots

In this section, you will plot the mass fraction (or wetness) of the condensed phase. From the mass fraction and droplet number, it is possible to derive a particle diameter, which you will also plot in this section. Notice that the particle diameters appear in the flow at very small sizes, in the range of 5.0E-9 m, but grow rapidly so that, when leaving the nucleation zone, they are approximately an order of magnitude larger. Since the pressure is dropping through the turbine, droplet sizes continue to increase along with the wetness. To emphasize the particle diameters proceeding out of the nucleation zone, you will set the contour range for viewing. In flow regions near walls, where expansion rate is reduced, small amounts of liquid may have droplet diameters that grow to large sizes relative to those coming from the nucleation zone. Without setting the range, these particle diameters near the walls will be emphasized.

1. Create a contour plot showing the mass fraction of the condensed phase on `Turbo Surface 1`.

   Condensed droplets grow in size and accumulate mass at the expense of the gas phase.

   1. Create a new contour plot named `Mass Fraction of Condensed Phase H2Ol`.
   2. In the **Details** view on the **Geometry** tab, set **Locations** to `Turbo Surface 1` and **Variable** to `H2Ol.Mass Fraction`.
   3. Click **Apply**.

      Comparing the mass fraction in the non-equilibrium case to the equilibrium solution previously viewed, you can see that phase transition is considerably delayed such that it occurs in the blade passages of the rotor rather than the stator. This is a typical consequence of non-equilibrium flow, and reflects real flow situations.

2. Turn off the visibility of **Mass Fraction of Condensed Phase H2Ol** when you have finished observing the results.
3. Create a particle diameter contour plot on `Turbo Surface 1`.

   The size of the condensed droplets is calculated from the droplet number and volume fraction of the condensed phase.

   1. Create a new contour plot named `Particle Diameter`.
   2. In the **Details** view on the **Geometry** tab, set **Locations** to `Turbo Surface 1` and **Variable** to `H2Ol.Particle Diameter`.
   3. Set **Range** to `User Specified` then set **Min** to `0 [m]` and **Max** to `1e-07 [m]`.
   4. Click **Apply**.

4. Turn off the visibility of `Particle Diameter` when you have finished observing the results.

26.10.1.4. Gas and Condensed Phase Static Temperature Contour Plots

In a non-equilibrium prediction, the gas phase temperature is different from the condensed phase temperature. The former is determined from a transport equation and the latter from an algebraic relationship relating droplet temperature to its diameter (through a small droplet model already described).

In this section, you will plot the different temperature fields. It should be noted that, for the case of the condensed phase temperature field, before droplets are actually formed, there is no meaningful droplet temperature. At the point droplets are formed by nucleation, their temperature is at the gas phase. Once the droplets have grown in size, their temperature is very close to the saturation temperature. Because the non-equilibrium solution considers the condensed phase temperatures separate from the gas phase, the influence of thermodynamic losses are included in the predictions. This is because it becomes possible to account for heat flow between the vapor and condensed phases as they pass through the domain. Due to this, non-equilibrium efficiency predictions are more accurate than ones obtained using an equilibrium model.

1. Create a gas phase static temperature contour plot on `Turbo Surface 1`:

   1. Create a new contour plot named `Gas Phase Static Temperature`.
   2. In the **Details** view on the **Geometry** tab, set **Locations** to `Turbo Surface 1` and **Variable** to `H2Og.Temperature`.
   3. Click **Apply**.

2. Turn off the visibility of **Gas Phase Static Temperature** when you have finished observing the results.
3. Create a condensed phase static temperature contour plot on `Turbo Surface 1`.
    1. Create a new contour plot named `Condensed Phase Static Temperature`.
    2. In the **Details** view on the **Geometry** tab, set **Locations** to `Turbo Surface 1` and **Variable** to `H2Ol.Temperature`.
    3. Click **Apply**.
4. Once you have observed the results, save the state and exit CFD-Post.

Chapter 27: Modeling a Gear Pump using an Immersed Solid

This tutorial includes:

27.1. Tutorial Features

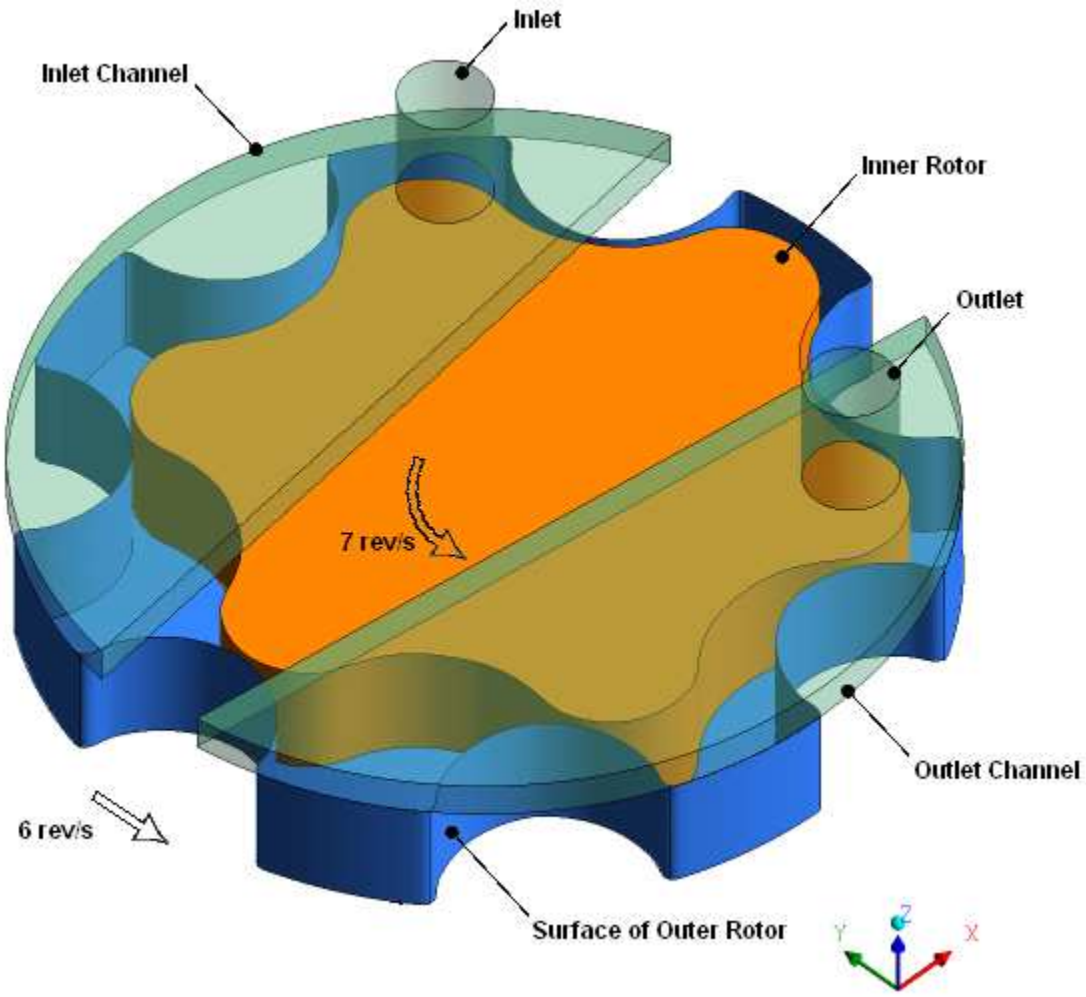
In this tutorial you will learn about:

- Setting up an immersed solids domain.
- Applying a counter-rotating wall boundary.
- Monitoring an expression during a solver run.
- Creating an XY-transient chart in CFD-Post.
- Creating a keyframe animation.

Component	Feature	Details
CFX-Pre	User Mode	General Mode
	Domain Type	Immersed Solid
		Fluid Domain
	Analysis Type	Transient
	Fluid Type	Continuous Fluid
	Boundary Conditions	Inlet Boundary
		Outlet Boundary
	Domain Interface	Fluid Fluid
CFD-Post	Chart	Mass Flow Rate
	Animation	Keyframe

27.2. Overview of the Problem to Solve

In this tutorial, you will simulate a gear pump that drives a flow of water. This tutorial makes use of the Immersed Solids capability of ANSYS CFX in order to model a solid that moves through a fluid. For more information on immersed solids see Immersed Solids.



The outlet has an average relative static pressure of 1 psi; the relative total pressure at the inlet is 0 psi. The inner rotor (gear) rotates at a rate of 7 revolutions per second; the outer rotor rotates at 6 revolutions per second. The diameter of the fluid region between the rotors is approximately 7.3 cm.

You will use an immersed solid domain to model the inner rotor, a rotating fluid domain to model the water immediately surrounding the inner rotor, and a stationary fluid domain to model the water in the inlet and outlet channels. To model the stationary pump housing (not shown in the figure), you will apply a counter-rotating wall condition to the top (high Z) surface of the rotating fluid domain, on the non-overlap portion (which lies between the inlet and outlet channels). To model the upper surfaces of the teeth of the outer rotor, you will apply a rotating wall condition on the non-overlap portions of the lower (low Z) surfaces of the inlet and outlet chambers. For more information about non-overlap conditions, see Non-overlap Boundary Conditions in the *CFX-Solver Modeling Guide*.

The following conditions will be met to promote the establishment of a cyclic flow pattern:

- The mesh of the rotating domain should be rotationally periodic so that it looks the same after each (outer) rotor tooth passes.

- The mesh on the outer boundary of the immersed solid domain should be rotationally periodic so that it looks the same after each (inner) rotor tooth passes. (The mesh inside the immersed solid domain has no effect in this tutorial.)
- An integer number of time steps should pass as one rotor tooth passes.

27.3. Before You Begin

It is strongly recommended that you complete the previous tutorials before trying this one. However, if this is the first tutorial you are working with, it is important to review the following topics before beginning:

- *Preparing a Working Directory and Starting ANSYS CFX in Standalone Mode* (p. 1)
- *Running ANSYS CFX Tutorials in ANSYS Workbench* (p. 2)
- *Changing the Display Colors* (p. 5)
- *Playing a Tutorial Session File* (p. 4)

27.4. Starting CFX-Pre

1. Prepare the working directory using the following files in the `examples` directory:
   - `ImmersedSolid.pre`
   - `ImmersedSolid.gtm`
2. Set the working directory and start CFX-Pre.

   For details, see *Preparing a Working Directory and Starting ANSYS CFX in Standalone Mode* (p. 1)

27.5. Defining a Case in CFX-Pre

If you want to set up the case automatically using a tutorial session file, run `ImmersedSolid.pre`. For details, see *Playing a Tutorial Session File* (p. 4). Then proceed to *Obtaining a Solution Using CFX-Solver Manager* (p. 492).

If you want to set up the case manually, proceed to the following steps:

This section describes the step-by-step definition of the flow physics in CFX-Pre for a steady-state simulation.

1. In CFX-Pre, select **File** > **New Case**.
2. Select **General** and click **OK**.
3. Select **File** > **Save Case As**.
4. Under **File name**, type `ImmersedSolid.cfx`.
5. Click **Save**.

27.6. Importing the Mesh

1. Edit `Case Options` > `General` in the **Outline** tree view and ensure that **Automatic Default Domain** is turned off.

   Default domain generation should be turned off because you will create three domains manually later in this tutorial.
2. Click **OK**.

3. Select **File** > **Import** > **Mesh**.
4. In the **Import Mesh** dialog box, set **Files of type** to `CFX Mesh (*gtm *cfx)` and select `Immersed-Solid.gtm` from your working directory.
5. Click **Open**.

27.7. Creating Expressions for Time Step and Total Time

Next, you will create an expression defining the time step size for this transient analysis. One tooth of the inner (or outer) rotor passes every 1/42 s. Choose a time step that resolves this motion in 30 intervals.

1. From the main menu, select **Insert** > **Expressions, Functions and Variables** > **Expression**.
2. In the **Insert Expression** dialog box, type `dt`.
3. Click **OK**.
4. Set **Definition**, to `(1/42)[s]/30`.
5. Click **Apply** to create the expression.

Next, you will create an expression defining the total simulation time. Make the simulation run long enough for 3 rotor teeth to pass: 3/42 s. This will give the solution time to establish a periodic nature.

1. Create an expression called `total time`.
2. Set **Definition** to `(3/42)[s]`.
3. Click **Apply**.

27.8. Setting the Analysis Type

Define the simulation as transient, using the expressions you created earlier.

1. Under the **Outline** tab, edit *Analysis Type*  .
2. Apply the following settings:

Tab	Setting	Value
Basic Settings	External Solver Coupling > Option	None
	Analysis Type > Option	Transient
	Analysis Type > Time Duration > Option	Total Time
	Analysis Type > Time Duration > Total Time	total time [1 (p. 485)]
	Analysis Type > Time Steps > Option	Timesteps
	Analysis Type > Time Steps > Timesteps	dt
	Analysis Type > Initial Time > Option	Automatic with Value
	Analysis Type > Initial Time > Time	0 [s]

Footnote

1. You first need to click the *Enter Expression*  icon beside the field.

3. Click **OK**.

27.9. Creating the Domains

This simulation requires three domains: two fluid domains and one immersed solid domain. First you will create an immersed solid domain.

27.9.1. Creating an Immersed Solid Domain

Ensure that no default domain is present under `Flow Analysis 1`. If a default domain is present, right-click it and select **Delete**.

Create the immersed solid domain as follows:

1. Select **Insert** > **Domain** from the main menu, or click *Domain*  .
2. In the **Insert Domain** dialog box, set the name to `ImmersedSolid` and click **OK**.
3. Apply the following settings:

Tab	Setting	Value
Basic Settings	Location and Type > Location	Inner Rotor
	Location and Type > Domain Type	Immersed Solid
	Location and Type > Coordinate Frame	Coord 0
	Domain Models > Domain Motion > Option	Rotating
	Domain Models > Domain Motion > Angular Velocity	7 [rev s^-1]
	Domain Models > Domain Motion > Axis Definition > Option	Two Points
	Domain Models > Domain Motion > Axis Definition > Rotation Axis From	0.00383, 0, 0
	Domain Models > Domain Motion > Axis Definition > Rotation Axis To	0.00383, 0, 1

4. Click **OK**.

27.9.2. Creating the Stationary Fluid Domain

Create the stationary fluid domain according to the problem description:

1. Create a new domain named `StationaryFluid`.
2. Apply the following settings:

Tab	Setting	Value
Basic Settings	Location and Type > Location	Channels
	Location and Type > Domain Type	Fluid Domain
	Location and Type > Coordinate Frame	Coord 0
	Fluid and Particle Definitions	Fluid 1
	Fluid and Particle Definitions > Fluid 1 > Option	Material Library
	Fluid and Particle Definitions > Fluid 1 > Material	Water
	Fluid and Particle Definitions > Fluid 1 > Morphology > Option	Continuous Fluid
	Domain Models > Pressure > Reference Pressure	0 [psi]
	Domain Models > Buoyancy > Option	Non Buoyant
	Domain Models > Domain Motion > Option	Stationary
	Domain Models > Mesh Deformation > Option	None
Fluid Models	Heat Transfer > Option	None
	Turbulence > Option	k-Epsilon
	Turbulence > Wall Function	Scalable
	Combustion > Option	None
	Thermal Radiation > Option	None
Initialization	Domain Initialization	(Selected)
	Domain Initialization > Initial Conditions > Velocity Type	Cartesian
	Domain Initialization > Initial Conditions > Cartesian Velocity Components > Option	Automatic with Value
	Domain Initialization > Initial Conditions > Cartesian Velocity Components > U	0 [m s^-1]
	Domain Initialization > Initial Conditions > Cartesian Velocity Components > V	0 [m s^-1]
	Domain Initialization > Initial Conditions > Cartesian Velocity Components > W	0 [m s^-1]

Tab	Setting	Value
	Domain Initialization > Initial Conditions > Static Pressure > Option	Automatic with Value
	Domain Initialization > Initial Conditions > Static Pressure > Relative Pressure	0 [psi]
	Domain Initialization > Initial Conditions > Turbulence > Option	Medium (Intensity = 5%)

3. Click **OK**.

27.9.3. Creating the Rotating Fluid Domain

Create the rotating fluid domain according to the problem description:

1. From the **Outline** tree view, right-click `Simulation` > `Flow Analysis 1` > `StationaryFluid` and select **Duplicate**.
2. Right-click `Simulation` > `Flow Analysis 1` > `Copy of StationaryFluid` and select **Rename**.
3. Rename the domain to `RotatingFluid`.
4. Edit `RotatingFluid`.
5. Apply the following settings:

Tab	Setting	Value
Basic Settings	Location and Type > Location	Gear Chamber
	Domain Models > Domain Motion > Option	Rotating
	Domain Models > Domain Motion > Angular Velocity	6 [rev s^-1]
	Domain Models > Domain Motion > Axis Definition > Option	Coordinate Axis
	Domain Models > Domain Motion > Axis Definition > Rotation Axis	Global Z

6. Click **OK**.

27.10. Creating the Domain Interface

Add a domain interface that connects the `StationaryFluid` and `RotatingFluid` domains:

1. Click **Insert** > **Domain Interface** from the main menu or click *Domain Interface*  .
2. Accept the default domain interface name and click **OK**.
3. Apply the following settings:

Tab	Setting	Value
Basic Settings	Interface Type	Fluid Fluid
	Interface Side 1 > Domain (Filter)	StationaryFluid
	Interface Side 1 > Region List	Channel Side
	Interface Side 2 > Domain (Filter)	RotatingFluid
	Interface Side 2 > Region List	Chamber Side
	Interface Models > Option	General Connection
	Interface Models > Frame Change/Mixing Model > Option	Transient Rotor Stator
	Interface Models > Pitch Change > Option[1 (p. 488)]	None
	Mesh Connection Method > Mesh Connection > Option	GGI
Additional Interface Models	Mass and Momentum > Option	Conservative Interface Flux
	Mass and Momentum > Interface Model > Option	None

Footnote

1. Setting this option to `None` will generate a global warning in the message window below the viewer. In this case, the warning can be ignored because the full 360° are being modeled on both sides of the interface.

4. Click **OK**.

Apply a counter-rotating no-slip wall condition to the non-overlap portion of the domain interface on the rotating domain side, because this surface represents part of the stationary housing of the pump.

1. Edit `RotatingFluid` > `Domain Interface 1 Side 2`.

   If this object does not appear in the tree view, then edit `Case Options` > `General`, select **Show Interface Boundaries in Outline Tree**, and click **OK**.

2. Apply the following settings:

Tab	Setting	Value
Nonoverlap Conditions	Nonoverlap Conditions	(Selected)
	Nonoverlap Conditions > Mass and Momentum > Option	No Slip Wall
	Nonoverlap Conditions > Mass and Momentum > Wall Velocity	(Selected)
	Nonoverlap Conditions > Mass and Momentum > Wall Velocity > Option	Counter Rotating Wall

3. Click **OK**.

Apply a rotating no-slip wall condition to the non-overlap portions of the domain interface on the stationary domain side, because these surfaces represent faces of the rotor teeth of the outer rotor, and the latter rotates at 6 rev/s about the Z axis.

1. Edit `StationaryFluid` > `Domain Interface 1 Side 1`.
2. Apply the following settings:

Tab	Setting	Value
Nonoverlap Conditions	Nonoverlap Conditions	(Selected)
	Nonoverlap Conditions > Mass and Momentum > Option	No Slip Wall
	Nonoverlap Conditions > Mass and Momentum > Wall Velocity	(Selected)
	Nonoverlap Conditions > Mass and Momentum > Wall Velocity > Option	Rotating Wall
	Nonoverlap Conditions > Mass and Momentum > Wall Velocity > Angular Velocity	6 [rev s^-1]
	Nonoverlap Conditions > Mass and Momentum > Wall Velocity > Axis Definition > Option	Coordinate Axis
	Nonoverlap Conditions > Mass and Momentum > Wall Velocity > Axis Definition > Rotation Axis	Global Z

3. Click **OK**.

27.11. Creating Boundary Conditions

This section outlines the steps to create the inlet and outlet boundary conditions, as specified in the problem description.

27.11.1. Inlet Boundary

Create a total pressure inlet at a relative pressure of 0 psi:

1. In the **Outline** tree view, right-click `StationaryFluid` and select **Insert** > **Boundary**.
2. Set **Name** to `in` and click **OK**.
3. Apply the following settings:

Tab	Setting	Value
Basic Settings	Boundary Type	Inlet
	Location	Inlet
Boundary Details	Mass And Momentum > Option	Total Pressure (stable)

Tab	Setting	Value
	Mass And Momentum > Relative Pressure	0 [psi]
	Flow Direction > Option	Normal to Boundary Condition
	Turbulence > Option	Medium (Intensity = 5%)

4. Click **OK**.

27.11.2. Outlet Boundary

Create an outlet with a relative average static pressure of 1 psi:

1. Create a boundary named `out` in the `StationaryFluid` domain.
2. Apply the following settings:

Tab	Setting	Value
Basic Settings	Boundary Type	Outlet
	Location	Outlet
Boundary Details	Mass And Momentum > Option	Average Static Pressure
	Mass And Momentum > Relative Pressure	1 [psi]
	Mass And Momentum > Pres. Profile Blend	0.05
	Pressure Averaging > Option	Average Over Whole Outlet

3. Click **OK**.

27.12. Setting Solver Control

1. Click *Solver Control*  .
2. Apply the following settings:

Tab	Setting	Value
Basic Settings	Advection Scheme > Option	High Resolution
	Transient Scheme > Option	Second Order Backward Euler
	Transient Scheme > Timestep Initialization > Option	Automatic
	Turbulence Numerics > Option	First Order
	Convergence Control > Min. Coeff. Loops	1
	Convergence Control > Max. Coeff. Loops	10

Tab	Setting	Value
	Convergence Control > Fluid Timescale Control > Timescale Control	Coefficient Loops
	Convergence Criteria > Residual Type	RMS
	Convergence Criteria > Residual Target	1.0 E −4

3. Click **OK**.

27.13. Setting Output Control

Set up the solver to output transient results files that record pressure, velocity, and velocity in the stationary frame, on every time step:

1. Click *Output Control*  .
2. Click the **Trn Results** tab.
3. In the **Transient Results** list box, click *Add new item*  , set **Name** to `Transient Results 1`, and click **OK**.
4. Apply the following settings to `Transient Results 1`:

Setting	Value
Option	Selected Variables
File Compression	Default
Output Variables List	Pressure, Velocity, Velocity in Stn Frame
Output Boundary Flows	(Selected)
Output Boundary Flows > Boundary Flows	All
Output Frequency > Option	Every Timestep

5. Click the **Monitor** tab.
6. Select **Monitor Objects**.
7. Under **Monitor Points and Expressions**:
    1. Click *Add new item*  .
    2. Accept the default name and click **OK**.
    3. Set **Option** to `Expression`.
    4. Set **Expression Value** to `massFlow()@in`.
8. Click **OK**.

27.14. Writing the CFX-Solver Input (.def) File

1. Click *Define Run*  .
2. Apply the following settings:

Setting	Value
File name	ImmersedSolid.def

3. Click **Save**.

Note

A warning message will appear due to the global warning that was mentioned earlier in *Creating the Domain Interface* (p. 487). Click **Yes**.

CFX-Solver Manager automatically starts and, on the **Define Run** dialog box, the **Solver Input File** is set.

4. If using Standalone Mode, quit CFX-Pre, saving the simulation (`.cfx`) file at your discretion.

27.15. Obtaining a Solution Using CFX-Solver Manager

Note

Due to sensitivity to initial conditions, this problem cannot be run in parallel and therefore must be run in serial.

When CFX-Pre has shut down and the CFX-Solver Manager has started, obtain a solution to the CFD problem by following the instructions below:

1. In CFX-Solver Manager, ensure that the **Define Run** dialog box is displayed.

   If CFX-Solver Manager is launched from CFX-Pre, the information required to perform a solver run is entered automatically in the **Define Run** dialog box.

2. Click **Start Run**.

   The solver run begins and the progress is displayed in a split screen.

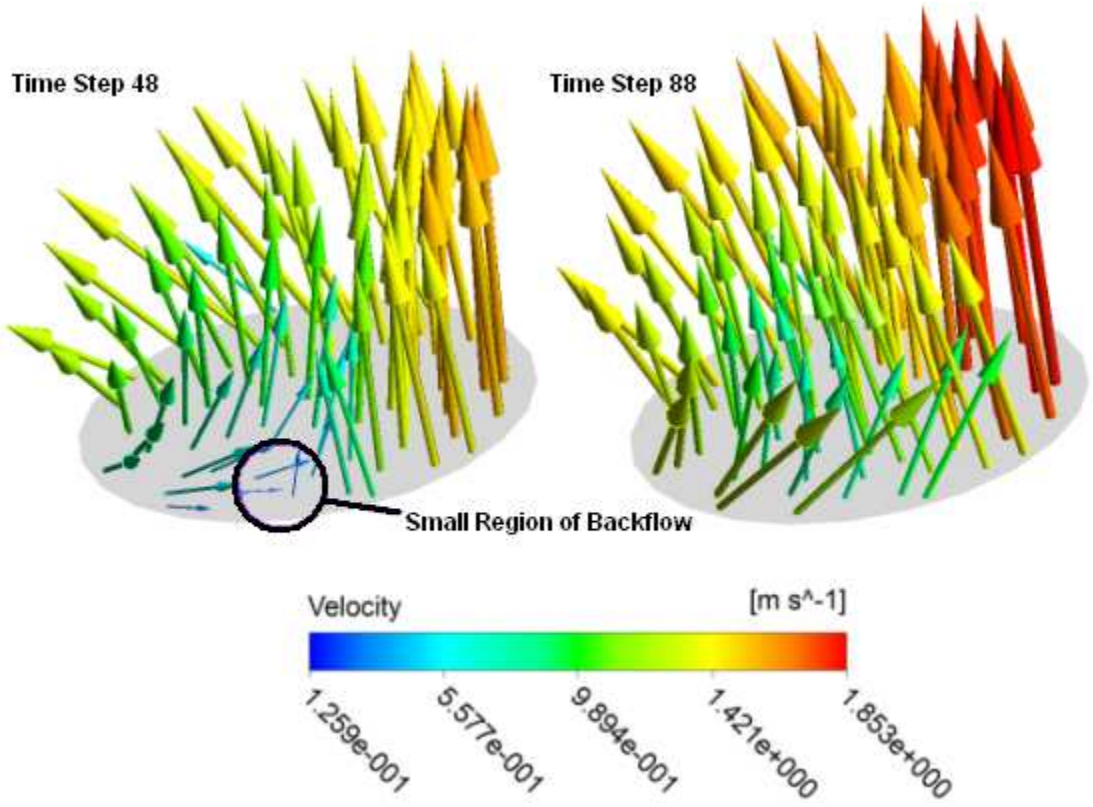
3. Click the **User Points** tab (which appears after the first time step has been computed) and monitor the value of `Monitor Point 1` as the solution proceeds.
4. Rescale the monitor plot so that you can readily see the time-periodic oscillations in mass flow that occur after the initial transient phase:
   1. Right-click anywhere in the **User Points** plot and select **Monitor Properties**.
   2. In the **Monitor Properties: User Points** dialog box, on the **Range Settings** tab, select **Set Manual Scale (Linear)**.
   3. Set the lower and upper bounds to 0.015 and 0.055 respectively.
   4. Click **OK**.

5. Select the check box next to **Post-Process Results** when the completion message appears at the end of the run.
6. If using Standalone Mode, select the check box next to **Shut down CFX-Solver Manager**.
7. Click **OK**.

Note

During the Solver Manager run, you may observe a notice at the 47th and 48th time steps warning you that "A wall has been placed at portion(s) of an OUTLET boundary condition ... to prevent fluid from flowing into the domain." The mass flow at the inlet drops to its lowest level throughout the cycle at this point, causing a reduction in the velocity at the outlet. Because there is turbulence at the outlet, this reduced velocity allows a tiny vortex to produce a small, virtually negligible, amount of backflow at the outlet. *Figure 27.1* (p. 493) shows velocity vectors at the outlet when the mass flow at the inlet is lowest (48th time step) and when the mass flow is greatest (88th time step). In the figure, you can see where this slight backflow occurs for the 48th time step.

Figure 27.1 Velocity Vectors on the Outlet



27.16. Viewing the Results in CFD-Post

In this section, you will generate a chart to show the mass flow rate through the machine as a function of time. You will also prepare an animation of the machine in operation, complete with velocity vectors.

27.16.1. Creating a Chart of Mass Flow versus Time

During the solver run, you observed a monitor plot that showed mass flow versus time step. Here, you will make a similar plot of mass flow versus time. As you did in the monitor plot, adjust the vertical axis range to focus on the time-periodic oscillations in mass flow that occur after the initial transient phase.

1. When CFD-Post starts, the **Domain Selector** dialog box might appear. If it does, ensure that both the ImmersedSolid, RotatingFluid and StationaryFluid domains are selected, then click **OK** to load the results from these domains.
2. A dialog box will notify you that the case contains immersed solid domain. Click **OK** to continue.
3. Create a new chart named `Mass Flow Rate`.

   The **Chart Viewer** tab appears.
4. Apply the following settings:

Tab	Setting	Value
General	Type	XY-Transient or Sequence
	Title	Mass Flow Rate at the Inlet over Time

5. Click the **Data Series** tab.
6. If the **Data Series** list box is empty, right-click in it and select **New**, or click *New*  .
7. Apply the following settings:

Tab	Setting	Value
Data Series	Series 1	(Selected)
	Name	Inlet Mass Flow
	Data Source > Expression	(Selected)
	Data Source > Expression	massFlow()@in [1 (p. 494)]
Y Axis	Axis Range > Determine ranges automatically	(Cleared)
	Axis Range > Min	0.015
	Axis Range > Max	0.055

Footnote

1. You will have to either type it manually or right-click and choose **Functions** > **CFD-Post** > **massFlow()@** from the shortcut menu, then type `in`.

8. Click **Apply**.

The mass flow rate settles into a repeating pattern with a period of 1/42 s, which is the time it takes a rotor tooth to pass.

27.16.2. Creating a Velocity Vector Plot

Create a slice plane and then make a vector plot on the slice plane as follows:

1. Click the **3D Viewer** tab.
2. Create a new plane named `Plane 1`.
3. Apply the following settings:

Tab	Setting	Value
Geometry	Domains	RotatingFluid
	Definition > Method	XY Plane
	Definition > Z	0.003 [m]

4. Click **Apply**.
5. Turn off the visibility of `Plane 1`.
6. Create a new vector plot named `Vector 1`.
7. Apply the following settings:

Tab	Setting	Value
Geometry	Domains	All Domains
	Definition > Locations	Plane 1
	Definition > Sampling	Rectangular Grid
	Definition > Spacing	0.03
	Definition > Variable	Velocity
Color	Mode	Use Plot Variable
	Range	User Specified
	Min	0 [m s^-1]
	Max	0.8 [m s^-1]
Symbol	Symbol	Arrow3D
	Symbol Size	15
	Normalized Symbols	(Cleared)

8. Click **Apply**.

27.16.3. Changing the Appearance in Preparation for an Animation

Make the inlet and outlet visible as follows:

1. Edit `StationaryFluid` > `in`.
2. Apply the following settings:

Tab	Setting	Value
Render	Show Faces	(Cleared)
	Show Mesh Lines	(Selected)

Tab	Setting	Value
	Show Mesh Lines > Edge Angle	105 [degree]
	Show Mesh Lines > Line Width	2
	Show Mesh Lines > Color Mode	Default

3. Click **Apply**.
4. Apply the same settings to `StationaryFluid` > `out`.

Make the inlet and outlet channels visible as follows:

1. Edit `StationaryFluid` > `StationaryFluid Default`.
2. Apply the following settings:

Tab	Setting	Value
Render	Show Faces	(Selected)
	Show Faces > Transparency	0.8
	Show Mesh Lines	(Cleared)

3. Click **Apply**.

Make the walls of the rotating fluid domain visible as follows:

1. Edit `RotatingFluid` > `RotatingFluid Default`.
2. Apply the following settings:

Tab	Setting	Value
Color	Mode	Constant
	Color	(White)
Render	Show Faces	(Selected)
	Show Faces > Transparency	0.0
	Show Mesh Lines	(Cleared)

3. Click **Apply**.

Make the walls of the immersed solid domain visible as follows:

1. Edit `ImmersedSolid` > `ImmersedSolid Default`.
2. Apply the following settings:

Tab	Setting	Value
Color	Mode	Constant
	Color	(Blue)
Render	Show Faces	(Selected)
	Show Faces > Transparency	0.0
	Show Mesh Lines	(Cleared)

3. Click **Apply**.

Make the following other changes in preparation for the animation that you will create in the next section:

1. Right-click a blank area in the viewer and select **Predefined Camera** > **View From +Z**.
2. Rotate the view a few degrees so that you can see the 3D nature of the geometry.
3. Turn off the visibility of `User Locations and Plots` > `Wireframe`.

27.16.4. Creating a Keyframe Animation

In this section, you will generate an animation that shows the changing velocity field on `Plane 1`. To take advantage of the periodic nature of the solution, you will record a short animation that can be played in a repeating loop in an MPEG player. Start the animation at the 61st time step (a time at which the flow has settled into a repeating pattern) and end it at the 90th time step. The 60th time step corresponds with 2/42 s, and the 90th corresponds with 3/42 s; the 1/42 s interval is the period over which the solution repeats. Because the 60th and 90th time steps look the same, the 60th time step is omitted to avoid having a pair of adjacent identical frames in the animation when the latter is played in a repeating loop.

1. Click *Timestep Selector* and load the 61st time step.
2. Click *Animation* .
3. In the **Animation** dialog box, select the **Keyframe Animation** option.
4. Click *New* to create `KeyframeNo1`.
5. Select `KeyframeNo1`, then set **# of Frames** to 28, then press **Enter** while the cursor is in the **# of Frames** box.

> **Tip**
>
> Be sure to press **Enter** and confirm that the new number appears in the list before continuing.

6. Use the **Timestep Selector** to load the 90th time step.
7. In the **Animation** dialog box, click *New* to create `KeyframeNo2`.
8. Ensure that *More Animation Options* is pushed down to show more animation settings.
9. Select **Loop**.
10. Ensure that the *Repeat forever* button next to **Repeat** is not selected (not pushed down).
11. Select **Save Movie**.
12. Set **Format** to `MPEG1`.
13. Click *Browse* next to **Save Movie**.
14. Set **File name** to `ImmersedSolid.mpg`.
15. If required, set the path location to a different directory.
16. Click **Save**.

The movie file name (including path) has been set, but the animation has not yet been produced.

17. Click *To Beginning*  .

    This ensures that the animation will begin at the first keyframe.

18. After the first keyframe has been loaded, click *Play the animation*  .

    - The MPEG will be created as the animation proceeds.
    - This will be slow, since results for each time step will be loaded and objects will be created.
    - To view the movie file, you need to use a viewer that supports the MPEG format.

    Note

    To explore additional animation options, click the **Options** button. On the **Advanced** tab of the **Animation Options** dialog box, there is a **Save Frames As Image Files** check box. By selecting this check box, the JPEG or PPM files used to encode each frame of the movie will persist after movie creation; otherwise, they will be deleted.

19. When you have finished, quit CFD-Post.

Chapter 28: Drop Curve for Cavitating Flow in a Pump

This tutorial includes:

28.1. Tutorial Features

In this tutorial you will learn about:

- Preparing and running a series of related simulations to generate cavitation performance data for a pump.
- Creating a drop curve chart in CFD-Post.
- Using isosurfaces in CFD-Post to visualize regions of cavitation.

Component	Feature	Details
CFX-Pre	User Mode	General Mode
	Analysis Type	Steady State
	Fluid Type	Water at 25 C
		Water Vapour at 25 C
	Fluid Models	Homogeneous Model
	Domain Type	Single Domain
	Turbulence Model	k-Epsilon
	Heat Transfer	Isothermal
	Boundary Conditions	Inlet (Subsonic)
		Outlet (Subsonic)
		Wall (Counter Rotating)
	Timestep	Physical Time Scale
CFD-Post	Plots	Contour

28.2. Overview of the Problem to Solve

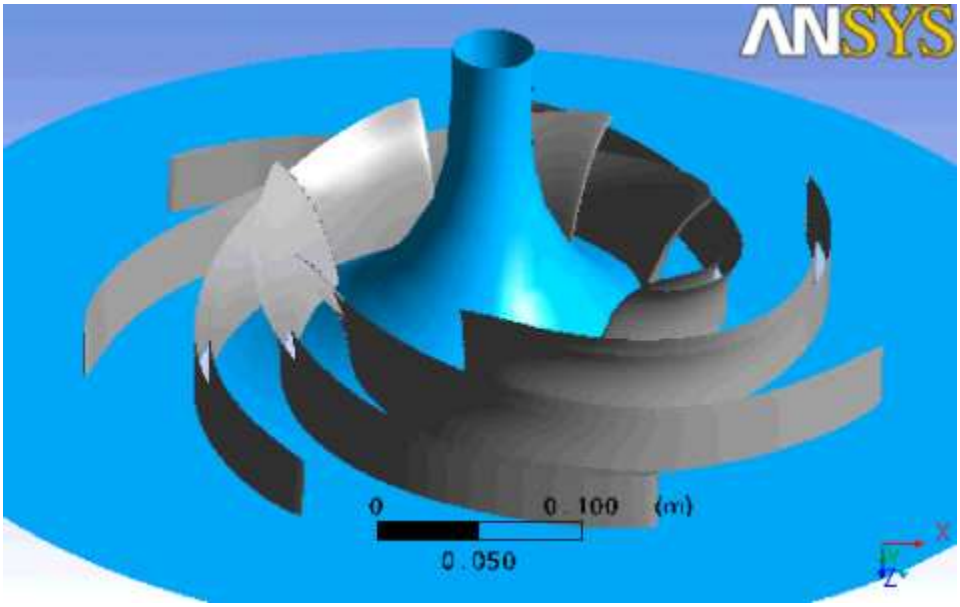
This tutorial uses a simple pump to illustrate the basic concepts of setting up, running and postprocessing a cavitation problem in ANSYS CFX.

When liquid is suddenly accelerated in order to move around an obstruction, a decrease in the local pressure is present. Sometimes, this pressure decrease is substantial enough that the pressure falls below the saturation

pressure determined by the temperature of the liquid. In such cases, the fluid begins to vaporize in a process called cavitation. Cavitation involves a very rapid increase in the volume occupied by a given mass of fluid, and when significant, can influence the flow distribution and operating performance of the device. In addition, the vaporization of the liquid, and the subsequent collapse of the vapor bubbles as the local pressure recovers, can cause damage to solid surfaces. For these reasons (among others), it is desirable, in the design and operation of devices required to move liquid, to be able to determine if cavitation is present, including where and the extent of the cavitation. Furthermore it is also useful to examine this behavior for a range of conditions. For details please refer to the "CFX Best Practices Guide for Cavitation".

The model conditions for this example are turbulent and incompressible. The speed and direction of rotation of the pump is 132 rad/sec about the Z-axis (positive rotation following the right hand rule). The relevant problem parameters are:

- Inflow total pressure = 100000 Pa
- Outflow mass flow = 16 kg/s
- Inlet turbulence intensity = 0.03
- Inlet length scale = 0.03 m



The SHF (Societe Hydraulique Francaise) pump has seven impeller blades. Due to the periodic nature of the geometry, only a single blade passage of the original pump needs to be modeled, thus minimizing the computer resources required to obtain a solution.

The objective of this tutorial is to show pump cavitation performance in the form of a drop curve. The drop curve is a chart of Head versus Net Positive Suction Head (NPSH). This tutorial provides the data for the drop curve, but also has instructions for optionally generating the data by running a series of simulations with progressively lower inlet pressures. Each simulation is initialized with the results of the previous simulation.

28.3. Before You Begin

If this is the first tutorial you are working with, it is important to review the following topics before beginning:

- *Preparing a Working Directory and Starting ANSYS CFX in Standalone Mode* (p. 1)
- *Running ANSYS CFX Tutorials in ANSYS Workbench* (p. 2)

- *Changing the Display Colors* (p. 5)
- *Playing a Tutorial Session File* (p. 4)

28.4. Starting CFX-Pre

1. Prepare the working directory using the following files in the `examples` directory:
   - `CavitationIni.pre`
   - `CavitationIni.cfx`
   - `Cavitation_100000.pre`
   - `Cavitation.gtm`
2. Set the working directory and start CFX-Pre.

   For details, see *Preparing a Working Directory and Starting ANSYS CFX in Standalone Mode* (p. 1)

28.5. High Pressure Simulation of Pump without Cavitation Model

A steady-state high-pressure (inlet pressure of 100,000 Pa) simulation of the pump without cavitation (that is, simulation of the pump without water vapor) will first be set up to be used as an initial values file for the cavitation simulation later on in the tutorial.

If you want to set up the simulation automatically using a tutorial session file, run `CavitationIni.pre`. For details, see *Playing a Tutorial Session File* (p. 4). Then proceed to *Obtaining a Solution Using CFX-Solver Manager* (p. 507).

If you want to set up the simulation manually, proceed to the following steps:

1. In CFX-Pre, select **File** > **New Case**.
2. Select **General** and click **OK**.
3. Select **File** > **Save Case As**.
4. Under **File name**, type `CavitationIni.cfx`.
5. Click **Save**.
6. Choose to overwrite `CavitationIni.cfx` if the software asks you for confirmation.

28.5.1. Importing the Mesh

1. Right-click `Mesh` and select **Import Mesh** > **Other**.

   The **Import Mesh** dialog box appears.
2. Apply the following settings

Setting	Value
Files of type	CFX Mesh (*gtm *cfx)
File name	Cavitation.gtm

3. Click **Open**.

28.5.2. Loading Materials

Since this tutorial uses water at 25 °C and water vapor at 25 °C, you need to load these materials. Note that you will only use the liquid water for the first part of the tutorial. The vapor is being loaded now in anticipation of using it for the cavitation model later in the tutorial.

1. In the **Outline** tree view, right-click `Simulation` > `Materials` and select **Import Library Data**.

   The **Select Library Data to Import** dialog box is displayed.
2. Expand `Water Data`.
3. Select both `Water at 25 C` and `Water Vapour at 25 C`, holding the **Ctrl** key.
4. Click **OK**.

28.5.3. Creating the Domain

1. Edit `Case Options` > `General` in the **Outline** tree view and ensure that **Automatic Default Domain** is turned on.

   A domain named `Default Domain` should appear under the `Simulation` branch.
2. Rename `Default Domain` to `Pump`.
3. Edit `Pump`.
4. Under **Fluid and Particle Definitions**, delete `Fluid 1` and create a new fluid definition named `Liquid Water`.
5. Apply the following settings:

Tab	Setting	Value
Basic Settings	Fluid and Particle Definitions	Liquid Water
	Fluid and Particle Definitions > Liquid Water > Material	Water at 25 C [1 (p. 502)]
	Domain Models > Pressure > Reference Pressure	0 [atm]
	Domain Models > Domain Motion > Option	Rotating
	Domain Models > Domain Motion > Angular Velocity	132 [radian s^-1]

Footnote

1. Click the *Ellipsis* icon ... to open the **Material** dialog box.

6. Click **OK**.

28.5.4. Creating the Boundaries

28.5.4.1. Inlet Boundary

1. Create a new boundary named `Inlet`.
2. Apply the following settings:

Tab	Setting	Value
Basic Settings	Boundary Type	Inlet
	Location	INBlock INFLOW
Boundary Details	Mass And Momentum > Option	Stat. Frame Tot. Press
	Mass And Momentum > Relative Pressure	100000 [Pa]
	Flow Direction > Option	Cartesian Components
	Flow Direction > X Component	0
	Flow Direction > Y Component	0
	Flow Direction > Z Component	1
	Turbulence > Option	Intensity and Length Scale
	Turbulence > Option > Fractional Intensity	0.03
	Turbulence > Option > Eddy Length Scale	0.03 [m]

3. Click **OK**.

28.5.4.2. Outlet Boundary

1. Create a new boundary named `Outlet`.
2. Apply the following settings:

Tab	Setting	Value
Basic Settings	Boundary Type	Outlet
	Location	OUTBlock OUTFLOW
Boundary Details	Mass and Momentum > Option	Mass Flow Rate
	Mass and Momentum > Mass Flow Rate	16 [kg/s]

3. Click **OK**.

28.5.4.3. Wall Boundaries

Set up the hub and shroud to be a stationary (non-rotating) wall.

1. Create a new boundary named `Stationary Wall.`
2. Apply the following settings to `Stationary Wall`:

Tab	Setting	Value
Basic Settings	Boundary Type	Wall
	Location	OUTBlock HUB, OUTBlock SHROUD [1 (p. 504)]
Boundary Details	Mass and Momentum > Wall Velocity	(Selected)
	Mass and Momentum > Wall Velocity > Option	Counter Rotating Wall

Footnote

1. Click the *Ellipsis* icon ... to open the **Selection Dialog** dialog box. In that dialog box, select `OUTBlock HUB` and `OUTBlock SHROUD`, holding the **CTRL** key. Click **OK**.

3. Click **OK**.

28.5.5. Creating Domain Interfaces

1. Click **Insert** > **Domain Interface** and, in the dialog box that appears, set **Name** to `Periodic Interface` and click **OK**.
2. Apply the following settings to `Periodic Interface`:

Tab	Setting	Value
Basic Settings	Interface Side 1 > Region List	INBlock PER1, OUTBlock PER1, Passage PER1 [1 (p. 504)]
	Interface Side 2 > Region List	INBlock PER2, OUTBlock PER2, Passage PER2 [2 (p. 504)]
	Interface Models > Option	Rotational Periodicity

Footnotes

1. Click the *Ellipsis* icon ... to open the **Selection Dialog** dialog box. In that dialog box, select `INBlock PER1,OUTBlock PER1` and `Passage PER1`, holding the **CTRL** key. Click **OK**.
2. Click the *Ellipsis* icon ... to open the **Selection Dialog** dialog box. In that dialog box, select `INBlock PER2,OUTBlock PER2` and `Passage PER2`, holding the **CTRL** key. Click **OK**.

3. Click **OK**.

28.5.5.1. Inblock to Passage Interface

1. Select **Insert** > **Domain Interface** and in the dialog box that appears, set **Name** to `Inblock to Passage Interface` and click **OK**.
2. Apply the following settings to `Inblock to Passage Interface`:

Tab	Setting	Value
Basic Settings	Interface Side 1 > Region List	OUTFLOW INBlock
	Interface Side 2 > Region List	INFLOW Passage
	Mesh Connection Method > Mesh Connection > Option	1:1

3. Click **OK**.

28.5.5.2. Passage to Outblock Interface

1. Select **Insert** > **Domain Interface** and in the dialog box that appears, set the Name to `Passage to Outblock Interface` and click **OK**.
2. Apply the following settings to `Passage to Outblock Interface`:

Tab	Setting	Value
Basic Settings	Interface Side 1 > Region List	OUTFLOW Passage
	Interface Side 2 > Region List	INFLOW OUTBlock
	Mesh Connection Method > Mesh Connection > Option	1:1

3. Click **OK**.

With the boundary conditions and domain interfaces defined above, the default boundary of a rotating wall is applied to the blade and the upstream portions of the hub and shroud.

28.5.6. Setting Initial Values

The initial values that will be setup are consistent with the inlet boundary conditions settings.

1. Click *Global Initialization*  .
2. Apply the following settings:

Tab	Setting	Value
Global Settings	Initial Conditions > Cartesian Velocity Components > Option	Automatic with Value
	Initial Conditions > Cartesian Velocity Components > U	0 [m/s]
	Initial Conditions > Cartesian Velocity Components > V	0 [m/s]
	Initial Conditions > Cartesian Velocity Components > W	1 [m/s]

Tab	Setting	Value
	Initial Conditions > Static Pressure > Option	Automatic with Value
	Initial Conditions > Static Pressure > Relative Pressure	100000 [Pa]
	Initial Conditions > Turbulence > Option	Intensity and Length Scale
	Initial Conditions > Turbulence > Fractional Intensity > Option	Automatic with Value
	Initial Conditions > Turbulence > Fractional Intensity > Value	0.03
	Initial Conditions > Turbulence > Eddy Length Scale > Option	Automatic with Value
	Initial Conditions > Turbulence > Eddy Length Scale > Value	0.03 [m]

3. Click **OK**.

28.5.7. Setting Solver Control

1. Click *Solver Control*  .
2. Apply the following settings:

Tab	Setting	Value
Basic Settings	Convergence Control > Max Iterations	500
	Convergence Control > Fluid Timescale Control > Timescale Control	Physical Timescale
	Convergence Control > Fluid Timescale Control > Physical Timescale	1e-3[s] [1 (p. 506)]
	Convergence Criteria > Residual Target	7.5e-6

Footnote

1. The physical timescale that will be setup is derived from the rotational speed of the blades and the fact that there are 7 blades in the full machine.

3. Click **OK**.

28.5.8. Writing the CFX-Solver Input (.def) File

1. Click *Define Run*  .

2. Apply the following settings:

Setting	Value
File name	CavitationIni.def

3. Click **Save**.

   CFX-Solver Manager automatically starts and, on the **Define Run** dialog box, the **Solver Input File** is set.

4. Save the simulation.

28.5.9. Obtaining a Solution Using CFX-Solver Manager

CFX-Solver Manager should be running. You will be able to obtain a solution to the CFD problem by following the instructions below.

1. Ensure **Define Run** is displayed.
2. Click **Start Run**.

   You may see a notice about an artificial wall at the inlet. This notice indicates that the flow is trying to exit at the inlet. This can be ignored because the amount of reverse flow is very low.

   CFX-Solver runs and attempts to obtain a solution. This can take a long time depending on your system. Eventually a dialog box is displayed stating that the simulation has completed.

3. At the end of the run, on the completion message that appears, select **Post-Process Results**.
4. If using Standalone mode, select **Shut down CFX-Solver Manager**.
5. Click **OK**.

28.5.10. Post-Processing the Solution for High-Pressure Simulation of Pump without Cavitation Model

CFD-Post should be running.

This case is run with temperatures around 300 K. The vapor pressure of water at this temperature is around 3574 Pa. To confirm that water vapor or cavitation is not likely for this operating condition of the pump, an isosurface of pressure at 3574 Pa will be created.

Create an isosurface of pressure at 3574 [Pa]:

1. Select **Insert > Location > Isosurface** and accept the default name.
2. Apply the following settings in the **Details** view:

Tab	Setting	Value
Geometry	Definition > Variable	Pressure
	Definition > Value	3574 [Pa]

3. Click **Apply**.

Notice that the isosurface does not appear. There is no place in the blade passage where the pressure is equal to 3574 Pa, which implies that there is no water vapor.

4. Quit CFD-Post, saving the state at your discretion.

28.6. High Pressure Simulation of Pump with Cavitation Model

CFX-Pre should be running; start it if necessary.

The simulation will be modified in CFX-Pre to include water vapor and enable the cavitation model. Monitor points will also be defined to observe the Net Positive Suction Head (NSPH) and pressure head values.

If you want to set up the simulation automatically and continue to *Obtaining a Solution using CFX-Solver Manager* (p. 511), run `Cavitation_100000.pre`.

28.6.1. Defining the Simulation in CFX-Pre

The following topics are discussed:

- *Modifying the Domain and Boundary Conditions* (p. 508)
- *Creating Expressions* (p. 510)
- *Adding Monitor Points* (p. 510)
- *Writing the CFX-Solver Input (.def) File* (p. 510)

28.6.1.1. Modifying the Domain and Boundary Conditions

1. If CFX-Pre is not already running, start it.
2. Open `CavitationIni.cfx` and save it as `Cavitation_100000.cfx`.

   "100000" indicates the inlet pressure of the simulation.
3. Open `Pump` for editing.
4. In the **Fluid and Particle Definitions** section, click *Add new item*  and name it `Water Vapor`.
5. Apply the following settings:

Tab	Setting	Value
Basic Settings	Fluid and Particle Definitions	Liquid Water
	Fluid and Particle Definitions > Liquid Water > Material	Water at 25 C [1 (p. 509)]
	Fluid and Particle Definitions	Water Vapor
	Fluid and Particle Definitions > Water Vapor > Material	Water Vapour at 25 C
Fluid Models	Multiphase > Homogeneous Model	(Selected) [2 (p. 509)]
Fluid Pair Models	Fluid Pair > Liquid Water | Water Vapor > Mass Transfer > Option	Cavitation
	Fluid Pair > Liquid Water | Water Vapor > Mass Transfer > Cavitation > Option	Rayleigh Plesset

Tab	Setting	Value
	Fluid Pair > Liquid Water | Water Vapor > Mass Transfer > Cavitation > Mean Diameter	2e-6 [m]
	Fluid Pair > Liquid Water | Water Vapor > Mass Transfer > Cavitation > Saturation Pressure	(Selected)
	Fluid Pair > Liquid Water | Water Vapor > Mass Transfer > Cavitation > Saturation Pressure > Saturation Pressure	3574 [Pa] [3 (p. 509)]

Footnotes

1. Click the *Ellipsis* icon ... to open the **Material** dialog box, then click the *Import Library Data* icon to open the **Select Library Data to Import** dialog box. In that dialog box, expand `Water Data` in the tree, then multi-select `Water at 25 C` and `Water Vapour at 25 C` and click **OK**.
2. The homogeneous model will be selected because the interphase transfer rate is very large in the pump. This results in all fluids sharing a common flow field and turbulence.
3. The pressure for a corresponding saturation temperature at which the water in the pump will boil into its vapor phase is 3574 Pa.

6. Click **OK**.
7. Open `Inlet` for editing and apply the following changes:

   Note that you are setting the inlet up to be 100% liquid water, hence a volume fraction of 1. Consequently, the volume fraction of the vapor is set to 0.

Tab	Setting	Value
Fluid Values	Boundary Conditions	Water Vapor
	Boundary Conditions > Water Vapor > Volume Fraction > Volume Fraction	0
	Boundary Conditions	Liquid Water
	Boundary Conditions > Liquid Water > Volume Fraction > Volume Fraction	1

8. Click **OK**.
9. Open `Outlet` for editing.
10. On the **Boundary Details** tab, set **Mass and Momentum > Option** to `Bulk Mass Flow Rate` and **Mass Flow Rate** to `16 [kg s^-1]`.
11. Click **OK**.

28.6.1.2. Creating Expressions

Expressions defining the Net Positive Suction Head (NPSH) and Head are created in order to monitor their values as the inlet pressure is decreased. By monitoring these values a drop curve can be produced.

- Create the following expressions.

Name	Definition
Ptin	massFlowAve(Total Pressure in Stn Frame)@Inlet
Ptout	massFlowAve(Total Pressure in Stn Frame)@Outlet
Wden	996.82 [kg m^-3]
Head	(Ptout-Ptin)/(Wden*g)
NPSH	(Ptin- Pvap)/(Wden*g)
Pvap	3574 [Pa]

28.6.1.3. Adding Monitor Points

Two monitor points will be added to track the NPSH and head using the expressions created in the previous step.

1. Click *Output Control*  .
2. On the **Monitor** tab, select **Monitor Options** and click *Add new item*  .
3. Enter NPSH `Point` as the name of the monitor point then enter the following settings:

Tab	Setting	Value
Monitor	Monitor Options > Monitor Points and Expressions > NPSH Point > Option	Expression
	Monitor Options > Monitor Points and Expressions > NPSH Point > Expression Value	NPSH

4. Create a second monitor point named `Head Point` with the same parameters as the first, with the exception that **Expression Value** is set to `Head`.
5. Click **OK**.

28.6.1.4. Writing the CFX-Solver Input (.def) File

1. Click *Define Run*  .
2. Apply the following settings:

Setting	Value
File name	Cavitation_100000.def

3. Click **Save**.

   CFX-Solver Manager automatically starts and, on the **Define Run** dialog box, the **Solver Input File** is set.

4. Save the simulation.

28.6.2. Obtaining a Solution using CFX-Solver Manager

CFX-Solver Manager should be running. Obtain a solution to the CFD problem by following these instructions:

1. Ensure **Define Run** is displayed.
2. Select **Initial Values Specification**.
3. Select `CavitationIni_001.res` for the initial values file using the *Browse*  tool.
4. Click **Start Run**.

   You may see a notice about an artificial wall at the inlet. This notice indicates that the flow is trying to exit at the inlet. This can be ignored because the amount of reverse flow is very low.

   CFX-Solver runs and attempts to obtain a solution. This will not require many iterations or much time, depending on your system, because only minor changes have been made from the initial simulation. A dialog box is displayed stating that the simulation has completed.

5. At the end of the run, on the completion message that appears, select **Post-Process Results**.
6. If using Standalone mode, select **Shut down CFX-Solver Manager**.
7. Click **OK**.

28.6.3. Post-Processing the Solution for High-Pressure Simulation of Pump with Cavitation Model

You will create an isosurface to observe the volume fraction of water vapor at 25 °C. Note that the pressure below the threshold is the same as found earlier in *High Pressure Simulation of Pump without Cavitation Model* (p. 501).

Create an isosurface for the volume fraction of water vapor at 25 °C, at 0.1:

1. CFD-Post should be running; start it if necessary.
2. Click **Insert** > **Location** > **Isosurface** and accept the default name.
3. Apply the following settings in the **Details** view:

Tab	Setting	Value
Geometry	Definition > Variable	Water Vapor.Volume Fraction
	Definition > Value	0.1

4. Click **Apply**.

   Notice that the isosurface is clear. There is no water vapor at 25 °C in the blade passage for the simulation with cavitation because at an inlet total pressure of 100000Pa, the minimum static pressure in the model is above the vapor pressure.

5. Quit CFD-Post saving the state at your discretion.

28.7. Pump Simulation with Cavitation Model Over Range of Inlet Pressures

In order to construct a drop curve for this cavitation case, the inlet pressure must be decremented from its initial value of 100000 Pa to 17500 Pa, and the Head and NPSH values must be recorded for each simulation. The results are provided in *Table 28.1: Pump Performance Data* (p. 512).

Table 28.1 Pump Performance Data

Inlet Pressure Pa	NPSH m	Head m
100000	9.859e+00	3.537e+01
80000	7.813e+00	3.535e+01
60000	5.767e+00	3.535e+01
40000	3.721e+00	3.536e+01
30000	2.698e+00	3.538e+01
20000	1.675e+00	3.534e+01
18000	1.470e+00	3.528e+01
17500	1.419e+00	3.184e+01

Optionally, if you want to generate the data shown in *Table 28.1: Pump Performance Data* (p. 512), then follow the instructions in the following two sections (*Writing CFX-Solver Input (.def) Files for Lower Inlet Pressures* (p. 512) and *Obtaining Solutions using CFX-Solver Manager* (p. 513)). Those instructions involve running several simulations in order to obtain a set of results files. As a benefit to doing this, you will have the results files required to complete an optional postprocessing exercise at the end of this tutorial. This optional postprocessing exercise involves using isosurfaces to visualize the regions of cavitation, and visually comparing these isosurfaces between different results files.

If you want to use the provided table data to produce a drop curve, proceed to *Generating a Drop Curve* (p. 514).

28.7.1. Writing CFX-Solver Input (.def) Files for Lower Inlet Pressures

Produce a set of definition (`.def`) files for the simulation, with each definition file specifying a progressively lower value for the inlet pressure:

1. CFX-Pre should be running; start it if necessary.
2. Open `Cavitation_100000.cfx`.
3. Open `Simulation` > `Flow Analysis 1` > `Pump` > `Inlet`.
4. On the **Boundary Details** tab change **Mass and Momentum** > **Relative Pressure** to `80000 [Pa]`.
5. Click **OK**.
6. Right-click `Simulation` and select **Write Solver Input File**.
7. Set **File name** to `Cavitation_80000.def`.
8. Click **Save**.

9. Change the inlet pressure and save a corresponding CFX-Solver input file for each of the 6 other pressures: 60000 Pa, 40000 Pa, 30000 Pa, 20000 Pa, 18000 Pa, and 17500 Pa.

Note

There are other techniques for defining a set of related simulations. For example, you could use configuration control, as demonstrated in *Flow from a Circular Vent* (p. 93).

28.7.2. Obtaining Solutions using CFX-Solver Manager

Run each of the CFX-Solver input files that you created in the previous step:

1. Start CFX-Solver Manager if it is not already running.
2. Ensure **Define Run** is displayed.
3. Under **Solver Input File**, click *Browse*  and select `Cavitation_80000.def`.
4. Select **Initial Values Specification**.
5. Select `Cavitation_100000_001.res` for the initial values file using the *Browse*  tool.
6. Click **Start Run**.

   You may see a notice about an artificial wall at the inlet. This notice indicates that the flow is trying to exit at the inlet. This can be ignored because the amount of reverse flow is very low.

   CFX-Solver runs and attempts to obtain a solution. This can take a long time depending on your system. Eventually a dialog box is displayed stating that the simulation has completed.
7. When the completion message appears at the end of the run, click **OK** without selecting the option to post-process the results.
8. Repeat this process until you have run all the CFX-Solver input files for all 6 other inlet pressures: 60000 Pa, 40000 Pa, 30000 Pa, 20000 Pa, 18000 Pa, and 17500 Pa. The pump simulation with cavitation model at an inlet pressure of 17500 Pa will converge poorly because the performance of the pump is decreasing considerably around that pressure. Note that the initial values should be taken from the previously generated results (`.res`) file.

28.7.3. Viewing the Solution for the Pump Simulation with Cavitation Model

To see the pump performance, you will generate a drop curve to show the pump performance over a range of inlet pressures. After generating the drop curve, there is an optional exercise for visualizing the cavitation regions using isosurfaces.

The optional exercise of visualizing the cavitation regions requires the results files from the 60000 Pa, 40000 Pa, 20000 Pa, and 17500 Pa simulations. If you have not generated those results files and want to complete the optional exercise, then generate the results files by following the instructions in:

- *Writing CFX-Solver Input (.def) Files for Lower Inlet Pressures* (p. 512)
- *Obtaining Solutions using CFX-Solver Manager* (p. 513)

28.7.3.1. Generating a Drop Curve

To generate a drop curve, you will need the values for Net Positive Suction Head (NPSH) and head as the inlet pressure decreases. This data is provided in *Table 28.1: Pump Performance Data* (p. 512). If you want to use that data, proceed to *Creating a Table of the Head and NPSH Values* (p. 514). If you have chosen to run all of the simulations and have obtained all of the results files, you can obtain the drop curve data yourself by following the instructions in the *Creating a Head-versus-NPSH Chart (Optional Exercise)* (p. 516) section.

28.7.3.1.1. Creating a Table of the Head and NPSH Values

1. Start CFD-Post.
2. Click **Insert** > **Table** and set the name to `Drop Curve Values`.
3. Enter the values from *Table 28.1: Pump Performance Data* (p. 512) for the 8 inlet pressures in the table.

   Enter the NPSH values in the left column and the head values in the right column.
4. Click *Save Table*  , and apply the following settings:

Setting	Value
File name	Drop Curve Values
Files of type	Comma Separated Values — Excel Readable (*.csv)

5. Click **Save**.

28.7.3.1.2. Creating a Head-versus-NPSH Chart

1. Click **Insert** > **Chart**.
2. Set the name to `Drop Curve` and click **OK**.
3. Apply the following settings:

Tab	Setting	Value
General	Title	Drop Curve
Data Series	Data Source > File	(Selected)
	Data Source > File > Browse 	Drop Curve Values.csv [1 (p. 515)]
X Axis	Axis Range > Determine ranges automatically	(Cleared)
	Axis Range > Min	0
	Axis Range > Max	10
	Axis Labels > Use data for axis labels	(Cleared)
	Axis Labels > Custom Label	NPSH [m]
Y Axis	Axis Range > Determine ranges automatically	(Cleared)
	Axis Range > Min	0
	Axis Range > Max	45

Tab	Setting	Value
	Axis Labels > Use data for axis labels	(Cleared)
	Axis Labels > Custom Label	Head [m]
Line Display	Line Display > Symbols	Triangle

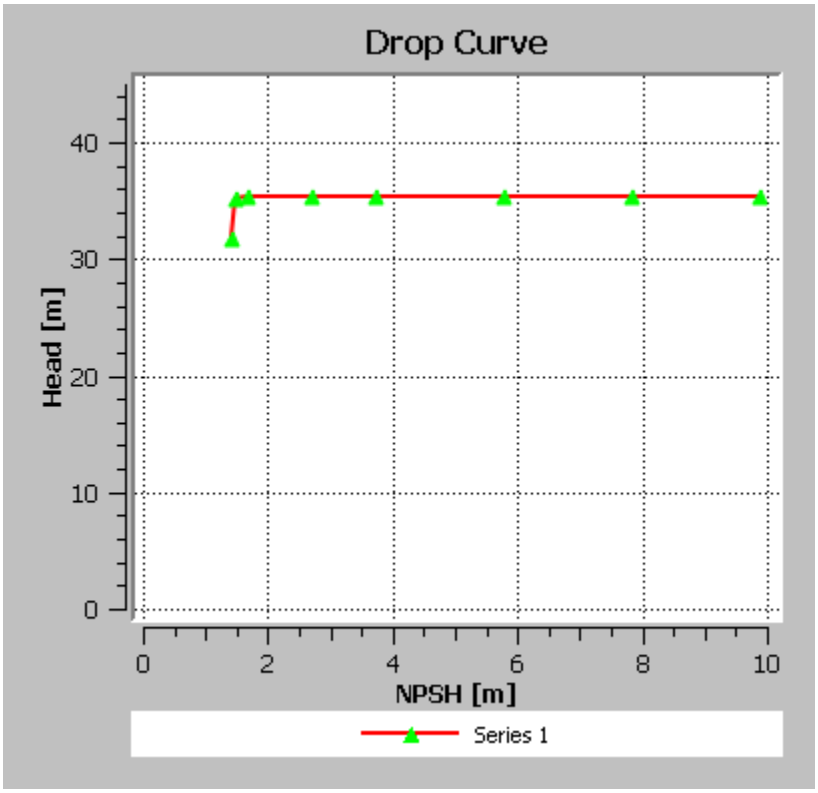
Footnote

1. Created in the previous steps.

4. Click **Apply** and proceed to *Viewing the Drop Curve* (p. 515).

28.7.3.1.3. Viewing the Drop Curve

Here is what the drop curve created in the earlier steps should look like:



You can see here that there is not significant degradation in the performance curve as the inlet total pressure is dropped. This is due to the fact that, for a part of the test, the inlet total pressure is sufficiently high to prevent cavitation, which implies that the normalized pressure rise across the pump is constant. Also, although you may start at a high inlet pressure where there is no cavitation, as you drop the inlet pressure, cavitation will appear but will have no significant impact on performance (incipient cavitation) until the blade passage has sufficient blockage due to vapor. At that point, performance degrades (rapidly in this case).

When the inlet total pressure reaches a sufficiently low value, cavitation occurs. The performance curve then starts to drop at a pressure of 18000 Pa as you can see on the chart. What is called the point of cavitation is often marked by the NPSH at which the pressure rise has fallen by a few percent, which is around 17500 Pa in this case.

If you want to complete an optional exercise on visualizing the cavitation regions, proceed to *Visualizing the Cavitation Regions (Optional Exercise)* (p. 517). Otherwise, quit CFD-Post, saving the state at your discretion.

28.7.3.1.4. Creating a Head-versus-NPSH Chart (Optional Exercise)

1. Start CFD-Post.
2. To load the results file, select **File** > **Load Results** or click *Load Results*  .
3. On the right side of the **Load Results File** dialog box, note down the current setting under **CFX run history and multi-configuration options**. Set this option to **Load complete history as:** > **A single case**, unless already set.

   Important

   This setting, under **CFX run history and multi-configuration options**, persists when you close CFD-Post. Ensure that you set this back to the original setting noted above, as instructed to do so at the end of the tutorial. Not doing so could lead to undesirable results when post-processing other cases.

4. In the **Load Results File** dialog box, select `Cavitation_17500_001.res`.
5. Click **Open**.

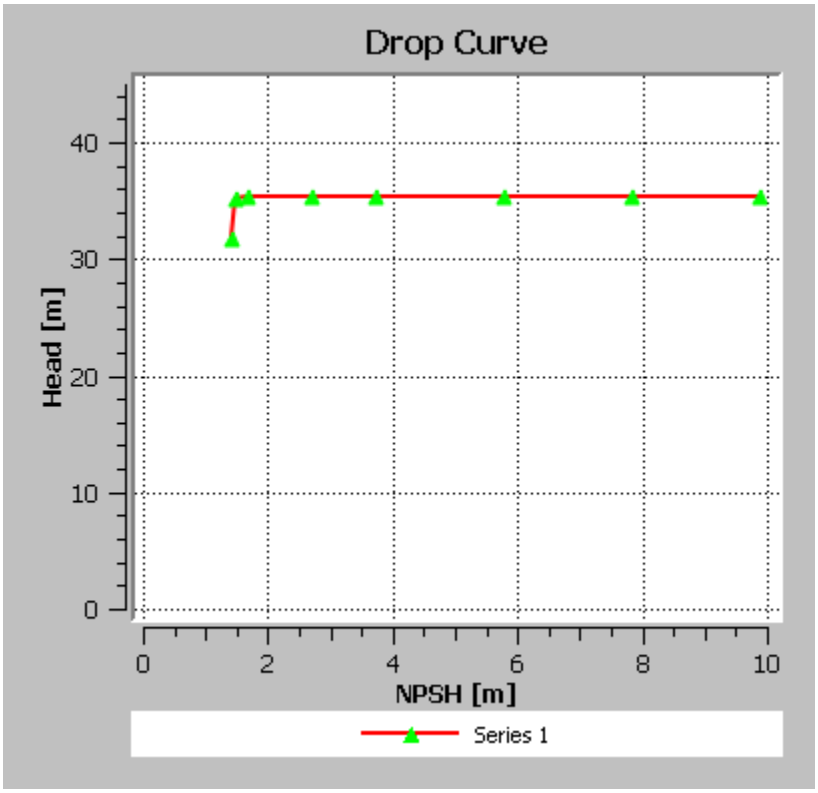
   When you started the CFX-Solver run using initial values, by default the**Continue History From** option was on. This enables the results file to retain a reference to the initial value results file. When the final results file is loaded into CFD-Post using the **Load complete history as: A single case**, it includes results from all the initial values files as well as the final results. Each of the previous initial values files is available as a timestep (in this case a sequence) through the **Timestep Selector**.
6. Click **OK** when prompted with a **Process Multiple Results as a Sequence** message.
7. Click **Insert** > **Chart** or click *Chart*  .
8. Set the name to `Drop Curve` and click **OK**.
9. Apply the following settings:

Tab	Setting	Value
General	Type	XY - Transient or Sequence
	Title	Drop Curve
Data Series	Data Source > Expression	(Selected)
	Data Source > Expression	Head
X Axis	Data Selection > Expression	NPSH
Y Axis	Axis Range > Determine ranges automatically	(Cleared)
	Axis Range > Min	0
	Axis Range > Max	45
Line Display	Line Display > Symbols	Triangle

10. Click **Apply** and proceed to *Viewing the Drop Curve* (p. 517).

28.7.3.1.5. Viewing the Drop Curve

Here is what the drop curve created in the earlier steps should look like:



You can see here that there is not significant degradation in the performance curve as the inlet total pressure is dropped. This is due to the fact that, for a part of the test, the inlet total pressure is sufficiently high to prevent cavitation, which implies that the normalized pressure rise across the pump is constant. Also, although you may start at a high inlet pressure where there is no cavitation, as you drop the inlet pressure, cavitation will appear but will have no significant impact on performance (incipient cavitation) until the blade passage has sufficient blockage due to vapor. At that point, performance degrades (rapidly in this case).

When the inlet total pressure reaches a sufficiently low value, cavitation occurs. The performance curve then starts to drop at a pressure of 18000 Pa as you can see on the chart. What is called the point of cavitation is often marked by the NPSH at which the pressure rise has fallen by a few percent, which is around 17500 Pa in this case.

Important

If you want to complete an optional exercise on visualizing the cavitation regions, proceed to *Visualizing the Cavitation Regions (Optional Exercise)* (p. 517). Otherwise, proceed to *Restoring CFX run history and multi-configuration options* (p. 518).

28.7.3.2. Visualizing the Cavitation Regions (Optional Exercise)

This is an optional part of the tutorial that requires the results files from the 60000 Pa, 40000 Pa, 20000 Pa, and 17500 Pa simulations. If you have not generated those results files and want to complete this optional exercise, then generate the results files by following the instructions in:

- *Writing CFX-Solver Input (.def) Files for Lower Inlet Pressures* (p. 512)

- *Obtaining Solutions using CFX-Solver Manager* (p. 513)

Cavitation does not occur for the 100000 Pa and 80000 Pa simulations. Create an isosurface for 10% water vapor (by volume fraction), for the 60000 Pa, 40000 Pa, 20000 Pa, and 17500 Pa simulations:

1. Use `Cavitation_60000_001.res` to create an isosurface:
   1. Launch CFD-Post and load `Cavitation_60000_001.res`.
   2. Select **Insert** > **Location** > **Isosurface** and accept the default name.
   3. Apply the following settings in the details view:

Tab	Setting	Value
Geometry	Definition > Variable	Water Vapor.Volume Fraction
	Definition > Value	0.1

   4. Click **Apply**.
2. Add `Cavitation_40000_001.res` to the current results:
   1. Select **File > Load Results**.
   2. Under **Case options**, select both **Keep current cases loaded** and **Open in new view**.
   3. Select `Cavitation_40000_001.res`.
   4. Click **Open**.
   5. Click a blank area inside the viewport named **View 2** (which contains the results that you just loaded) to make that viewport active, then turn on visibility for the isosurface in the **Outline** tree view.
3. In a similar way, load `Cavitation_20000_001.res` and `Cavitation_17500_001.res` and make the isosurface visible on these results.
4. Click *Synchronize camera in displayed views*  so that all viewports maintain the same camera position.
5. Rotate the view (from any viewport) to inspect the results.

   Observe that the amount of water vapor increases as the inlet pressure decreases.

Important

If you created the drop curve by setting the **CFX run history and multi-configuration options**, proceed to *Restoring CFX run history and multi-configuration options* (p. 518). Otherwise, quit CFD-Post, saving the state at your discretion.

28.7.3.3. Restoring CFX run history and multi-configuration options

As mentioned above the setting under **CFX run history and multi-configuration options** persists when you close CFD-Post. This section outlines the steps to restore **CFX run history and multi-configuration options** to its original setting.

1. Select **File** > **Close** to close the current file.
2. Click **Close** if prompted to save.

3. Load a results file by selecting **File** > **Load Results** or click *Load Results*  .
4. On the right side of the **Load Results File** dialog box, restore the original settings under **CFX run history and multi-configuration options**.
5. In the **Load Results File** dialog box, select `Cavitation_17500_001.res`.
6. Click **Open**.
7. Quit CFD-Post, by selecting **File** > **Quit**.

Chapter 29: Spray Dryer

This tutorial includes:

29.1. Tutorial Features

In this tutorial you will learn about:

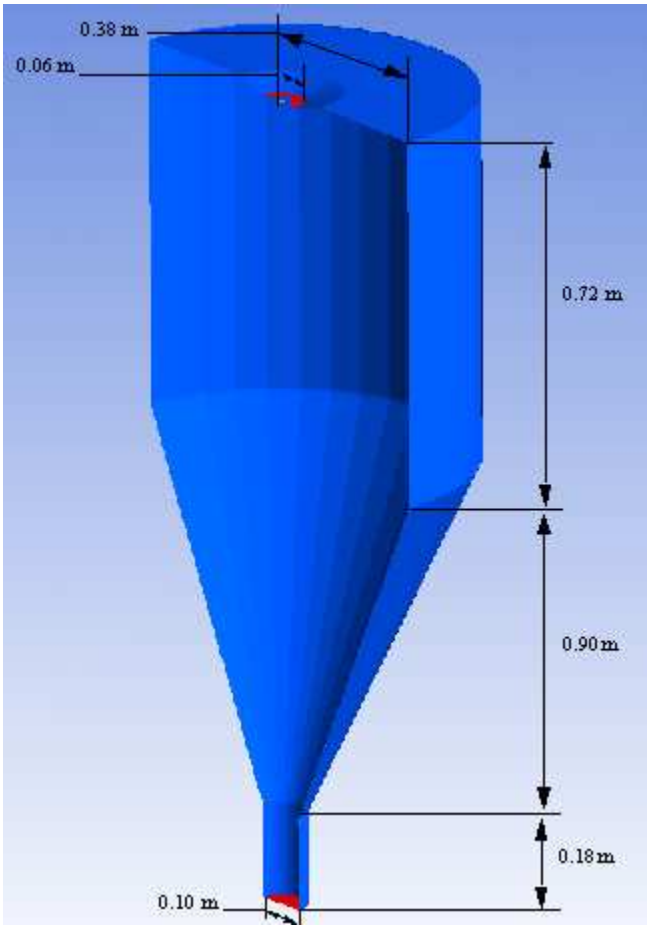
- Importing a CCL file in CFX-Pre.
- Editing and creating boundary conditions in CFX-Pre.
- Adding particles that evaporate.
- Creating a domain interface in CFX-Pre.
- Creating contour plots and inserting particle tracking in CFD-Post.

Component	Feature	Details
CFX-Pre	User Mode	General Mode
	Analysis Type	Steady State
	Fluid Type	General Fluid
	Domain Type	Single Domain
	Boundary Conditions	Water Nozzle
		Air Inlet
		Outlet
		Domain 1 Default
	Domain Interface	Fluid Fluid
	Timescale	Physical Timescale
	Particle Coupling Control	Selected
	Extra Output Variables List	Selected
CFD-Post	Plots	Contour Plots
		Particle Tracking

29.2. Overview of the Problem to Solve

In this example, a spray dryer is modeled in which water drops are evaporated by a hot air flow. The goal of this tutorial is to observe the variation of gas temperature and mass fraction of water vapor, and of averaged values of mean droplet diameter and droplet temperature in the spray dryer, as well as the temperature and size of individual water drops as they travel through the spray dryer.

The following figure shows approximately half of the full geometry. The spray dryer has two inlets named `Water Nozzle` and `Air Inlet`, and one outlet named `Outlet`. The `Water Nozzle` is where the liquid water enters in a primary air flow at a mass flow rate of 1.33e-4 kg/s. The `Air Inlet` is for the swirling, drying air flow. The `Water Nozzle` inlet is located in the middle of the circular `Air Inlet`. When the spray dryer is operating, the inlets are located at the top of the vessel and the outlet at the bottom.



Periodic boundaries are used to allow only a small section of the full geometry to be modeled. The geometry to be modeled consists of a 9 degree section of the axisymmetric dryer shape. The relevant parameters of this problem are:

- Static temperature at `Water Nozzle` = 300 K
- Size distribution for the drops being created by the `Water Nozzle` is prescribed using discrete diameter values and associated fractions of the droplet mass flow rate.
- `Air Inlet` mass and momentum axial component = 30 m/s (downwards along the axis of the spray dryer),`Air Inlet` mass and momentum radial component = 0 m/s, `Air Inlet` mass and momentum theta component = 10 m/s
- Static temperature at `Air Inlet` = 423 K

- Relative pressure at `Outlet` = 0 Pa
- Normal speed of Water = 10 m/s

The approach for solving this problem is to first import a CCL file with the fluid properties, domain and boundary conditions in CFX-Pre. Minor changes will be made to the information imported from the CCL file. Boundary conditions and a domain interface will also be added. In CFD-Post, contour plots will be created to see the variation of temperature, mass fraction of water, average mean particle diameter of liquid water, and averaged temperature of liquid water in the spray dryer. Finally, particle tracking will be used for plotting the temperature of liquid water.

29.3. Before You Begin

If this is the first tutorial you are working with, it is important to review the following topics before beginning:

- *Preparing a Working Directory and Starting ANSYS CFX in Standalone Mode* (p. 1)
- *Running ANSYS CFX Tutorials in ANSYS Workbench* (p. 2)
- *Changing the Display Colors* (p. 5)
- *Playing a Tutorial Session File* (p. 4)

29.4. Starting CFX-Pre

1. Prepare the working directory using the following files in the `examples` directory:
   - `SprayDryer.pre`
   - `spraydryer9.gtm`
2. In addition to the files from the `examples` directory, the `evaporating_drops.ccl` file from the `etc/model-templates` directory, should also be copied over to the working directory.
3. Set the working directory and start CFX-Pre.

   For details, see *Preparing a Working Directory and Starting ANSYS CFX in Standalone Mode* (p. 1)

29.5. Defining a Steady-State Simulation in CFX-Pre

If you want to set up the simulation automatically using a tutorial session file, run `SprayDryer.pre`. For details, see *Playing a Tutorial Session File* (p. 4). Then proceed to *Obtaining a Solution Using CFX-Solver Manager* (p. 530).

If you want to set up the simulation manually, proceed to the following steps:

This section describes the step-by-step definition of the flow physics in CFX-Pre for a steady-state simulation.

1. In CFX-Pre, select **File** > **New Case**.
2. Select **General** and click **OK**.
3. Edit `Case Options` > `General` in the **Outline** tree view and ensure that **Automatic Default Domain** is turned off.
4. Select **File** > **Save Case As**.
5. Under **File name**, type `SprayDryer`.
6. Click **Save**.

29.5.1. Importing the Mesh

1. Select **File** > **Import** > **Mesh**.
2. Apply the following settings

Setting	Value
Files of type	CFX Mesh (*gtm *cfx)
File name	spraydryer9.gtm

3. Click **Open**.

29.5.2. Importing the Evaporating CCL Drops Model Template

ANSYS CFX Command Language (CCL) consists of commands used to carry out actions in CFX-Pre, CFX-Solver Manager and CFD-Post. The physics for this simulation such as materials, domain and domain properties will be imported as a CCL. We will first analyze the `evaporating_drops.ccl` model template and then import it into the simulation.

Note

The physics for a simulation can be saved to a CCL (CFX Command Language) file at any time by selecting **File** > **Export** > **CCL**.

1. Open `evaporating_drops.ccl` with a text editor and take the time to look at the information it contains.

   The template sets up the materials water vapor H2O with a thermal conductivity of 193e-04 W/mK and water liquid H2Ol, which enters from the Water Nozzle. Note that the water data could also have been imported from the library in CFX-Pre. The template also creates a continuous gas phase named `Gas mixture` containing H2O and `Air Ideal Gas` and a binary mixture of H2O and H2Ol, which determines the rate of evaporation of the water. A domain named `Domain 1` that includes the `Gas mixture` and H2Ol as a fluid pair as well as an inlet boundary is also specified in the CCL file. The inlet boundary is set up with a default static temperature of 573 K.
2. Select **File** > **Import** > **CCL**

   The **Import CCL** dialog box appears.
3. Under **Import Method**, select **Append**. This will start with the existing CCL already generated and append the imported CCL.

Note

Replace is useful if you have defined physics and want to update or replace them with newly-imported physics.

4. Select `evaporating_drops.ccl`.
5. Click **Open**.

Note

An error message related to the parameter `Location` will appear in the message window. This error occurs as the CCL contains a location placeholder that is not part of the mesh. Ignore this error message as the issue will be addressed when `Domain 1` is being edited.

29.5.3. Editing the Domain

The fluid domain imported in the CCL file will be edited in this section.

1. In the tree view, right-click `Domain 1`, then click **Edit**.
2. Apply the following settings:

Tab	Setting	Value
Basic Settings	Location and Type > Location	B34
	Domain Models > Buoyancy > Option	Buoyant
	Domain Models > Buoyancy > Gravity X Dirn.	0.0 [m s^-1]
	Domain Models > Buoyancy > Gravity Y Dirn.	-9.81 [m s^-1]
	Domain Models > Buoyancy > Gravity Z Dirn.	0.0 [m s^-1]
	Domain Models > Buoyancy > Buoy. Ref. Density	1.2 [kg m^-3] [1 (p. 525)]
Fluid Specific Models	Fluid	Gas mixture
	Fluid > Gas mixture > Fluid Buoyancy Model > Option	Non Buoyant [2 (p. 525)]
	Fluid	H2Ol
	Fluid > H2Ol > Fluid Buoyancy Model > Option	Density Difference

Footnotes

1. The buoyancy reference density is set to 1.2 as representative of air.
2. Because any natural convection in the gas can be neglected, we can set the fluid to non buoyant.

3. Click **OK**.

29.5.4. Creating and Editing the Boundary Conditions

In this section, the `Inlet` and `Domain 1 Default` boundary conditions that were imported in the CCL file will be edited. Two boundary conditions, `Air Inlet` and `Outlet` will also be created for the spray dryer simulation.

29.5.4.1. Water Nozzle Boundary

The inlet boundary where the water enters in a primary air flow will be renamed and edited with the particle mass flow rate set consistent with the problem description. The particle diameter distribution will be set to `Discrete Diameter Distribution`, which will allow us to have particles of more than one specified diameter. Diameter values will be listed as specified in the problem description. A mass fraction as well as a number fraction will be specified for each of the diameter entries. The total of mass fractions and the total of number fractions will sum to unity.

1. In the tree view, under `Domain 1`, right-click `inlet`, then click **Rename**. Set the new name to `Water Nozzle`.
2. In the tree view, right-click `Water Nozzle`, then click **Edit**.
3. Apply the following settings:

Tab	Setting	Value
Basic Settings	Boundary Type	Inlet
	Location	two fluid nozzle
Boundary Details	Mass and Momentum > Option	Normal Speed
	Mass and Momentum > Normal Speed	10.0 [m s^-1]
	Heat Transfer > Option	Static Temperature
	Heat Transfer > Static Temperature	300.0 K
Fluid Values	H2Ol > Mass and Momentum > Option	Normal Speed
	H2Ol > Mass and Momentum > Normal Speed	10.0 [m s^-1]
	H2Ol > Particle Position > Number of Positions > Number	500 [1 (p. 527)]
	H2Ol > Particle Mass Flow > Mass Flow Rate	3.32e-6 [kg s^-1] [2 (p. 527)]
	H2Ol > Particle Diameter Distribution > Option	Discrete Diameter Distribution
	H2Ol > Particle Diameter Distribution > Diameter List	5.9e-6, 1.25e-5, 1.39e-5, 1.54e-5, 1.7e-5, 1.88e-5, 2.09e-5, 2.27e-5, 2.48e-5, 3.11e-5 [m]
	H2Ol > Particle Diameter Distribution > Mass Fraction List	10*0.1
	H2Ol > Particle Diameter Distribution > Number Fraction List	10*0.1
	H2Ol > Heat Transfer > Option	Static Temperature
	H2Ol > Heat Transfer > Static Temperature	300.0 K

Footnotes

1. The number of representative drops was chosen to be 500 through experience of particle transport calculations.
2. Note that this mass flow is only 1/40th of the total mass flow rate of water because only a 9 degree sector is modeled.

4. Click **OK**.

29.5.4.2. Air Inlet Boundary

A second inlet in which the swirling, drying air flow will enter will be created with temperature component, mass and momentum axial, radial and theta components set consistent with the problem description.

1. Select **Insert** > **Boundary** from the main menu or click *Boundary* .
2. Under **Name**, type `Air Inlet`.
3. Click **OK**.
4. Apply the following settings:

Tab	Setting	Value
Basic Settings	Boundary Type	Inlet
	Location	air inlet
Boundary Details	Mass and Momentum > Option	Cyl. Vel. Components
	Mass and Momentum > Axial Component	-30.0 [m s^-1]
	Mass and Momentum > Radial Component	0.0 [m s^-1]
	Mass and Momentum > Theta Component	10.0 [m s^-1]
	Axis Definition > Option	Coordinate Axis
	Axis Definition > Rotation Axis	Global Y
	Heat Transfer > Option	Static Temperature
	Heat Transfer > Static Temperature	423.0 K

5. Click **OK**.

29.5.4.3. Outlet Boundary

The outlet boundary will be created as an opening with pressure as specified in the problem description.

1. Select **Insert** > **Boundary** from the main menu or click *Boundary* .
2. Under **Name**, type `Outlet`.
3. Click **OK**.
4. Apply the following settings:

Tab	Setting	Value
Basic Settings	Boundary Type	Outlet
	Location	outlet
Boundary Details	Mass and Momentum > Option	Average Static Pressure
	Mass and Momentum > Relative Pressure	0.0 [Pa]

5. Click **OK**.

29.5.4.4. Domain 1 Default

The `Domain 1 Default` boundary will be edited to use a heat transfer coefficient of 3.0 [W m^-2 K^-1] and an outside temperature of 300 K.

1. In the tree view, right-click `Domain 1 Default`, then click **Edit**.
2. Apply the following settings:

Tab	Setting	Value
Boundary Details	Heat Transfer > Option	Heat Transfer Coefficient
	Heat Transfer > Heat Trans. Coeff.	3.0 [W m^-2 K^-1]
	Heat Transfer > Outside Temperature	300.0 [K]

3. Click **OK**.

29.5.5. Creating a Domain Interface

A domain interface will be created to connect the `Domain 1`, `periodic1` and `periodic 2` regions. The two sides of the periodic interface, `periodic1` and `periodic 2`, will be mapped by a single rotational transformation about an axis.

1. Select **Insert** > **Domain Interface**. Accept the default name.
2. Apply the following settings:

Tab	Setting	Value
Basic Settings	Interface Type	Fluid Fluid
	Interface Side 1 > Domain (filter)	Domain 1
	Interface Side 1 > Region List	periodic1
	Interface Side 2 > Domain (filter)	Domain 1
	Interface Side 2 > Region List	periodic 2
	Interface Models > Option	Rotational Periodicity
	Interface Models > Axis Definition > Rotation Axis	Global Y
	Mesh Connection Method > Mesh Connection > Option	Automatic

3. Click **OK**.

29.5.6. Setting Solver Control

1. Click *Solver Control*  .
2. Apply the following settings:

Tab	Setting	Value
Basic Settings	Convergence Control > Max. Iterations	100
	Convergence Control > Fluid Timescale Control > Timescale Control	Physical Timescale
	Convergence Control > Fluid Timescale Control > Physical Timescale	0.05 [s] [1 (p. 529)]
	Convergence Criteria > Residual Type	RMS
	Convergence Criteria > Residual Target	1.E-4

Footnote

1. Based on the air inlet speed and the size of the dryer.

3. Click **OK**.

29.5.7. Setting Output Control

In this section, two additional variables, H2O1.Averaged Mean Particle Diameter and H2O1.Averaged Temperature will be specified. These variables will be used when viewing the results in CFD-Post to understand the flow behavior.

1. Click *Output Control*  .
2. Apply the following settings:

Tab	Setting	Value
Results	Extra Output Variables List	Selected
	Extra Output Variables List > Extra Output Var. List	H2Ol.Averaged Mean Particle Diameter, H2Ol.Averaged Temperature [1 (p. 530)]

Footnote

1. Click the *Ellipsis* icon ... to open the **Extra Output Variable List** dialog box, then select `H2Ol.Averaged Mean Particle Diameter` and `H2Ol.Averaged Temperature,` holding the **Ctrl** key. Click **OK**.

3. Click **OK**.

29.5.8. Writing the CFX-Solver Input (.def) File

1. Click *Define Run*  .
2. Apply the following settings:

Setting	Value
File name	SprayDryer.def

3. Click **Save**.

   CFX-Solver Manager automatically starts and, on the **Define Run** dialog box, the **Solver Input File** is set.
4. Quit CFX-Pre, saving the simulation (`.cfx`) file.

29.6. Obtaining a Solution Using CFX-Solver Manager

When CFX-Pre has shut down and the CFX-Solver Manager has started, obtain a solution to the CFD problem by following the instructions below.

1. Ensure **Define Run** is displayed.

   CFX-Solver Input File should be set to `SprayDryer.def`.
2. Click **Start Run**.

   CFX-Solver runs and attempts to obtain a solution. This can take a long time depending on your system. Eventually a dialog box is displayed.
3. Select the check box next to **Post-Process Results** when the completion message appears at the end of the run.
4. If using Standalone Mode, select the check box next to **Shut down CFX-Solver Manager**.
5. Click **OK**.

29.7. Viewing the Results in CFD-Post

In this section, contour plots located on one of the periodic regions of the spray dryer will be created to illustrate the variation of temperature, water mass fraction, liquid water average mean particle diameter and liquid water averaged temperature. Finally, particle tracking will be used for plotting the temperature of liquid water. Particle tracking will trace the mean flow behavior in and around the complex geometry of the spray dryer.

29.7.1. Displaying the Temperature Using a Contour Plot

A contour plot located at the `Domain Interface 1 Side 1`, on the `Periodic 1` region, will first be created, and used to show the temperature variation through the spray dryer.

1. Right-click a blank area in the viewer and select **Predefined Camera** > **View From -Z**.

   This ensures that the view is set to a position that is best suited to display the results.
2. From the main menu, select **Insert** > **Contour**.
3. Set the name to `Temperature Contour`. Click **OK**.
4. Apply the following settings:

Tab	Setting	Value
Geometry	Location	Domain Interface 1 Side 1
	Variable	Temperature

5. Click **Apply**.
6. When you have finished, right-click the contour you just created in the tree view and select **Hide**.

29.7.2. Displaying the Water Mass Fraction Using a Contour Plot

A contour plot located at the `Periodic 1` region will be created and used to show the `H2O.Mass Fraction` variation through the spray dryer.

- Repeat steps 1-6 in the *Displaying the Temperature Using a Contour Plot* (p. 531) section. In step 3, change the contour name to `H2O Mass Fraction Contour`. In step 4, change the variable to `H2O.Mass Fraction`.

29.7.3. Displaying the Liquid Water Averaged Mean Particle Diameter Using a Contour Plot

A contour plot located at the `Periodic 1` region will be created and used to show the `H2Ol.Averaged Mean Particle Diameter` variation through the spray dryer.

- Repeat steps 1-6 in the *Displaying the Temperature Using a Contour Plot* (p. 531) section. In step 3, change the contour name to `H2Ol Averaged Mean Particle Diameter Contour`. In step 4, change the variable to `H2Ol.Averaged Mean Particle Diameter`.

29.7.4. Displaying the Liquid Water Averaged Temperature Using a Contour Plot

A contour plot located at the `Periodic 1` region will be created and used to show the `H2Ol.Averaged Temperature` variation through the spray dryer.

- Repeat steps 1-6 in the *Displaying the Temperature Using a Contour Plot* (p. 531) section. In step 3, change the contour name to `H2Ol Averaged Temperature Contour`. In step 4, change the variable to `H2Ol.Averaged Temperature`.

29.7.5. Displaying the Liquid Water Temperature Using Particle Tracking

This section outlines the steps for using particle tracking to trace the variation of the water temperature.

1. Right-click a blank area in the viewer and select **Predefined Camera** > **Isometric View (Z up)**.

   This ensures that the view is set to a position that is best suited to display the results.
2. From the main menu, select **Insert** > **Particle Track**.
3. Set the name to `H2Ol Temperature`. Click **OK**.
4. Apply the following settings:

Tab	Setting	Value
Color	Mode	Variable
	Variable	H20l.Temperature

5. Click **Apply**.

   From the contours and particle tracks, notice that the water droplets entering the spray dryer through the `Water Nozzle` recirculates in the region between the two inlets before merging in with the stream of hot air entering the spray dryer through the `Air Inlet`. Based on this flow of the water drop, the temperature of the hot gas coming from the `Air Inlet` decreases as the process takes place. During the spray drying cycle, the air transfers its thermal energy to the liquid water drops, leading to evaporation. As the air carries the thermal energy by convection, liquid water droplets that are close to the `Air Inlet` see their temperature increase, which leads to evaporation, resulting in a decrease in droplet diameter and an increase in the amount of water vapor.

29.7.6. Displaying the Diameter of a Water Drop Using Particle Tracking

This section outlines the steps for using particle tracking to trace the variation of water droplet diameter.

1. Repeat steps 2-5 in the *Displaying the Liquid Water Temperature Using Particle Tracking* (p. 532) section. In step 3, change the name to `H2Ol Mean Particle Diameter`. In step 4, change the variable name to `H20l.Mean Particle Diameter`

   From the water drop diameter particle track, we can see that as the air from the `Air Inlet` transfers its thermal energy to the liquid water, the diameter of water drops decreases as they evaporate. So when the water drop move away from the `Water Nozzle`, its diameter decreases as a function of the temperature increase.
2. Quit CFD-Post, saving the state (`.cst`) file at your discretion.

Chapter 30: Coal Combustion

This tutorial includes:

30.1. Tutorial Features

In this tutorial you will learn about:

- Importing a CCL file in CFX-Pre.
- Setting up and using Proximate/Ultimate analysis for hydrocarbon fuels in CFX-Pre.
- Viewing the results for nitrogen oxide in CFD-Post.

Component	Feature	Details
CFX-Pre	User Mode	General Mode
	Analysis Type	Steady State
	Fluid Type	Reacting Mixture, Hydrocarbon Fuel
	CCL File	Import
	Domain Type	Single Domain
	Boundaries	Coal Inlet
		Air Inlet
		Outlet
		No-slip Wall
		Periodic
		Symmetry
CFD-Post	Plots	Particle Tracking

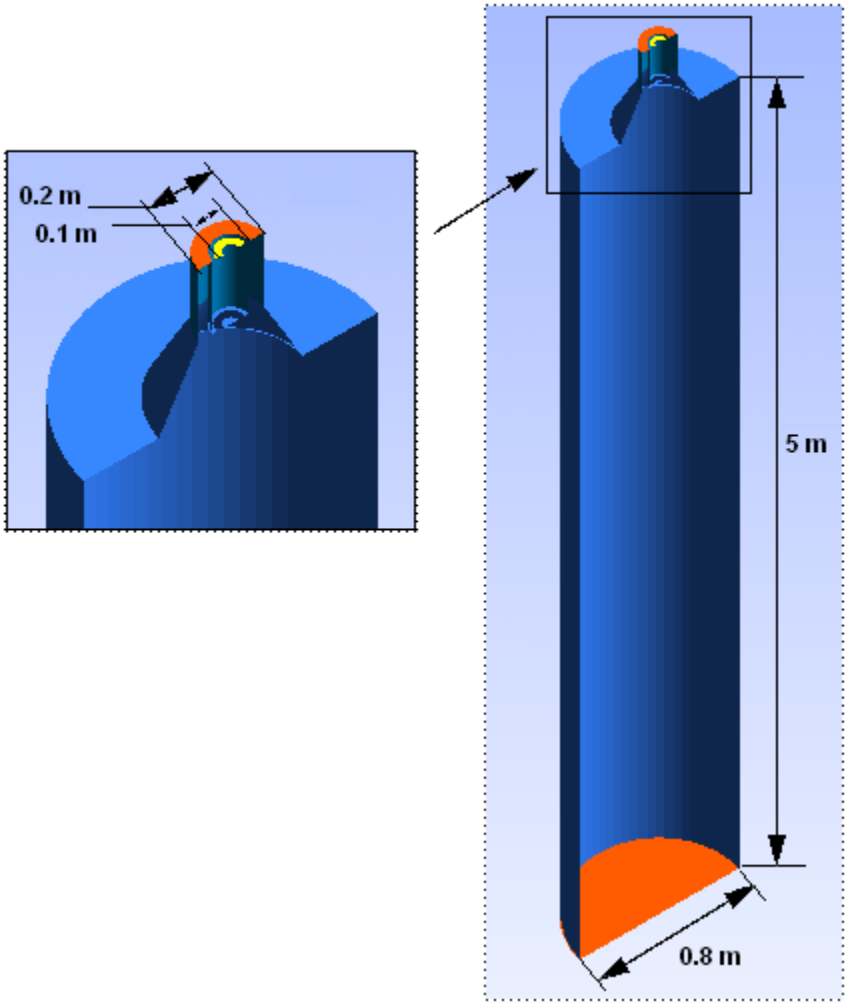
30.2. Overview of the Problem to Solve

In this tutorial, you will model coal combustion and radiation in a furnace. Three different coal combustion simulations will be set up:

- Coal Combustion with no-swirl burners where there is no release of nitrogen oxide during the burning process.

- Coal Combustion with swirl burners where there is no release of nitrogen oxide during the burning process.
- Coal Combustion with swirl burners where there is release of nitrogen oxide during the burning process.

The following figure shows half of the full geometry. The coal furnace has two inlets: `Coal Inlet` and `Air Inlet`, and one outlet. The `Coal Inlet` (see the inner yellow annulus shown in the figure inset) has air entering at a mass flow rate of 1.624e-3 kg/s and pulverized coal particles entering at a mass flow rate of 1.015e-3 kg/s. The `Air Inlet` (see the outer orange annulus shown in the figure inset) is where heated air enters the coal furnace at a mass flow rate of 1.035e-2 kg/s. The outlet is located at the opposite end of the furnace and is at a pressure of 1 atm.



The provided mesh occupies a 5 degree section of an axisymmetric coal furnace. Each simulation will make use of either symmetric or periodic boundaries to model the effects of the remainder of the furnace. In the case of non-swirling flow, a pair of symmetry boundaries is sufficient; in the case of flow with swirl, a periodic boundary with rotational periodicity is required.

The relevant parameters of this problem are:

- `Coal Inlet` static temperature = 343 K
- Size distribution for the drops being created by the `Coal Inlet` = 12, 38, 62, 88 μm
- `Air Inlet` static temperature = 573 K

- `Outlet` average static pressure = 0 Pa
- `Coal Gun` wall fixed temperature = 800 K
- `Coal Inlet` wall fixed temperature = 343 K
- `Air Inlet` wall fixed temperature = 573 K
- `Furnace wall` fixed temperature = 1400 K
- O2 mass fraction = 0.232
- Proximate/ultimate analysis data for the coal. Note that proximate/ultimate analysis data is used to characterize the properties of the coal including the content of moisture, volatile, free carbon, and ash, as well as the mass fractions of carbon, hydrogen and oxygen (the major components), sulfur and nitrogen.

The approach for solving this problem is to first import, into CFX-Pre, a CCL file with the proximate/ultimate analysis data for the coal and the required materials and reactions. The first simulation will be without nitrogen oxide or swirl. Only small changes to the boundary conditions will be made to create the second simulation, which has swirl in the flow. After each of the first two simulations, you will use CFD-Post to see the variation of temperature, water mass fraction and radiation intensity. You will examine particle tracks colored by temperature and by ash mass fraction. The last simulation has swirl and also involves the release of nitrogen oxide. Finally, you will use CFD-Post to see the distribution of nitrogen oxide in the third simulation.

30.3. Before You Begin

If this is the first tutorial you are working with, it is important to review the following topics before beginning:

- *Preparing a Working Directory and Starting ANSYS CFX in Standalone Mode* (p. 1)
- *Running ANSYS CFX Tutorials in ANSYS Workbench* (p. 2)
- *Changing the Display Colors* (p. 5)
- *Playing a Tutorial Session File* (p. 4)

30.4. Starting CFX-Pre

1. Prepare the working directory using the following files in the `examples` directory:
   - `CoalCombustion.gtm`
   - `CoalCombustion_Reactions_Materials.ccl`
   - `CoalCombustion_nonox.pre`
   - `CoalCombustion_nonox_swirl.pre`
   - `CoalCombustion_noxcpp_swirl.pre`
2. Set the working directory and start CFX-Pre.

   For details, see *Preparing a Working Directory and Starting ANSYS CFX in Standalone Mode* (p. 1)

30.5. Coal Combustion Simulation

If you want to set up the simulation automatically using a tutorial session file, run `CoalCombustion_nonox.pre`. For details, see *Playing a Tutorial Session File* (p. 4). Then proceed to *Obtaining a Solution for the Simulation using CFX-Solver Manager* (p. 547).

If you want to set up the simulation manually, proceed to the following steps:

You will first create a simulation where there is no release of nitrogen oxide, a hazardous chemical, during the process. Swirl burners will not be used in this simulation.

1. In CFX-Pre, select **File** > **New Case**.
2. Select **General** and click **OK**.
3. Edit `Case Options` > `General` in the **Outline** tree view and ensure that **Automatic Default Domain** and **Automatic Default Interfaces** are turned off.
4. Select **File** > **Save Case As**.
5. Set **File name** to `CoalCombustion_nonox.cfx`.
6. Click **Save**.

30.5.1. Importing the Mesh

1. Select **File** > **Import** > **Mesh**.
2. Set **Files of type** to `CFX Mesh(*gtm *cfx)`.
3. Select `CoalCombustion.gtm` from your working directory.
4. Click **Open**.
5. Right-click a blank area in the viewer and select **Predefined Camera** > **Isometric View (Z up)** from the shortcut menu.

30.5.2. Importing the Coal Combustion Materials CCL File

CFX Command Language (CCL) consists of commands used to carry out actions in CFX-Pre, CFX-Solver Manager, and CFD-Post. The proximate/ultimate analysis data for the coal as well as the materials and reactions required for the combustion simulation will be imported from the CCL file. You will review, then import, the contents of the `CoalCombustion_Reactions_Materials.ccl` file.

Note

The physics for a simulation can be saved to a CCL (CFX Command Language) file at any time by selecting **File** > **Export** > **CCL**.

1. Open `CoalCombustion_Reactions_Materials.ccl` with a text editor and take the time to look at the information it contains.

   The CCL sets up the following reactions:

   - `Fuel Gas Oxygen`
   - `HC Fuel Char Field`
   - `HC Fuel Devolat`
   - `Prompt NO Fuel Gas PDF`
   - `Thermal NO PDF`.

   The CCL also sets up the following materials:

   - `Ash`

- Char
- Fuel Gas
- Gas mixture
- HC Fuel
- HC Fuel Gas Binary Mixture
- Raw Combustible

The reactions Prompt NO Fuel Gas PDF and Thermal NO PDF are used only in the third simulation. Other pure substances required for the simulation will be loaded from the standard CFX-Pre materials library.

2. In CFX-Pre, select **File** > **Import** > **CCL**.

   The **Import CCL** dialog box appears.

3. Under **Import Method**, select **Replace**.

   This will replace the materials list in the current simulation with the ones in the newly imported CCL.

4. Under **Import Method**, select **Auto-load materials**.

   This will load pure materials such as CO2, H2O, N2, O2, and NO — the materials referenced by the imported mixtures and reactions — from the CFX-Pre materials library.

5. Select CoalCombustion_Reactions_Materials.ccl (the file you reviewed earlier).
6. Click **Open**.
7. Expand the Materials and Reactions branches under Simulation to make sure that all the materials and reactions described above are present.

30.5.3. Creating the Domain

Create a new domain named Furnace as follows:

1. Right-click Simulation > Flow Analysis 1 in the **Outline** tree view and click **Insert** > **Domain**.
2. Set **Name** to Furnace.
3. Click **OK**
4. On the **Basic Settings**, tab under **Fluid and Particle Definitions**, delete Fluid 1 and create a new fluid definition named Gas Mixture.
5. Click *Add new item*  and create a new fluid definition named HC Fuel.
6. Apply the following settings:

Tab	Setting	Value
Basic Settings	Location and Type > Location	B40
	Fluid and Particle Definitions	Gas Mixture
	Fluid and Particle Definitions > Gas Mixture > Material	Gas Mixture [1 (p. 539)]

Tab	Setting	Value
	Fluid and Particle Definitions > Gas Mixture > Morphology > Option	Continuous Fluid
	Fluid and Particle Definitions	HC Fuel
	Fluid and Particle Definitions > HC Fuel > Material	HC Fuel [2 (p. 539)]
	Fluid and Particle Definitions > HC Fuel > Morphology > Option	Particle Transport Solid
	Fluid and Particle Definitions > HC Fuel > Morphology > Particle Diameter Change	(Selected)
	Fluid and Particle Definitions > HC Fuel > Morphology > Particle Diameter Change > Option	Mass Equivalent[3 (p. 539)]
Fluid Models	Multiphase > Multiphase Reactions	(Selected)
	Multiphase > Multiphase Reactions > Reactions List	HC Fuel Char Field, HC Fuel Devolat
	Heat Transfer > Option	Fluid Dependent
	Combustion > Option	Fluid Dependent
	Thermal Radiation > Option	Fluid Dependent
Fluid Specific Models	Fluid	Gas Mixture
	Fluid > Gas Mixture > Heat Transfer > Heat Transfer > Option	Thermal Energy
	Fluid > Gas Mixture > Thermal Radiation > Option	Discrete Transfer
	Fluid > Gas Mixture > Thermal Radiation > Number of Rays	(Selected)
	Fluid > Gas Mixture > Thermal Radiation > Number of Rays > Number of Rays	32 [4 (p. 539)]
	Fluid	HC Fuel
	Fluid > HC Fuel > Heat Transfer > Heat Transfer > Option	Particle Temperature
Fluid Pair Models	Fluid Pair	Gas Mixture | HC Fuel
	Fluid Pair > Gas Mixture | HC Fuel > Particle Coupling	Fully Coupled
	Fluid Pair > Gas Mixture | HC Fuel > Momentum Transfer > Drag Force > Option	Schiller Naumann
	Fluid Pair > Gas Mixture | HC Fuel > Heat Transfer > Option	Ranz Marshall

Tab	Setting	Value
	Fluid Pair > Gas Mixture | HC Fuel > Thermal Radiation Transfer > Option	Opaque
	Fluid Pair > Gas Mixture | HC Fuel > Thermal Radiation Transfer > Emissivity	1 [5 (p. 539)]
	Fluid Pair > Gas Mixture | HC Fuel > Thermal Radiation Transfer > Particle Coupling	(Selected)
	Fluid Pair > Gas Mixture | HC Fuel > Thermal Radiation Transfer > Particle Coupling > Particle Coupling	Fully Coupled

Footnotes

1. Click the *Ellipsis* ... icon to open the **Material** dialog box, then select `Gas Mixture` under the `Gas Phase Combustion` branch. Click **OK**.
2. Click the *Ellipsis* ... icon to open the **Material** dialog box, then select `HC Fuel` under the `Particle Solids` branch. Click **OK**.
3. The use of the `Mass Equivalent` option for the particle diameter is used here for demonstration only. A physically more sensible setting for coal particles, which often stay the same size or get bigger during combustion, would be the use of the `Swelling Model` option with a **Swelling Factor** of `0.0` (the default) or larger.
4. Increasing the number of rays to 32 from the default 8, increases the number of rays leaving the bounding surfaces and increases the accuracy of the thermal radiation calculation.
5. With this setting, the particles are modeled as black bodies.

7. Click **OK**.

30.5.4. Creating the Boundary Conditions

In this section you will create boundary conditions for the coal inlet, the air inlet, the outlet, and multiple no-slip walls. You will also create two symmetry-plane boundary conditions for this no-swirl case.

30.5.4.1. Coal Inlet Boundary

You will create the coal inlet boundary with mass flow rate and static temperature set consistently with the problem description. The particle diameter distribution will be set to `Discrete Diameter Distribution` to model particles of more than one specified diameter. Diameter values will be listed as specified in the problem description. A mass fraction as well as a number fraction will be specified for each of the diameter entries. The total of mass fractions and the total of number fractions will sum to unity.

1. Create a boundary named `Coal Inlet`.

2. Apply the following settings to `Coal Inlet`:

Tab	Setting	Value
Basic Settings	Boundary Type	Inlet
	Location	CoalInlet
Boundary Details	Mass and Momentum > Option	Mass Flow Rate
	Mass and Momentum > Mass Flow Rate	0.001624 [kg s^-1]
	Flow Direction > Option	Normal to Boundary Condition
	Heat Transfer > Option	Static Temperature
	Heat Transfer > Static Temperature	343 [K]
	Component Details	O2
	Component Details > O2 > Option	Mass Fraction
	Component Details > O2 > Mass Fraction	0.232
Fluid Values	Boundary Conditions > HC Fuel > Particle Behavior > Define Particle Behavior	(Selected)
	Boundary Conditions > HC Fuel > Mass and Momentum > Option	Zero Slip Velocity
	Boundary Conditions > HC Fuel > Particle Position > Option	Uniform Injection
	Boundary Conditions > HC Fuel > Particle Position > Particle Locations	(Selected)
	Boundary Conditions > HC Fuel > Particle Position > Particle Locations > Particle Locations	Equally Spaced
	Boundary Conditions > HC Fuel > Particle Position > Number of Positions > Option	Direct Specification
	Boundary Conditions > HC Fuel > Particle Position > Number of Positions > Number	200
	Boundary Conditions > HC Fuel > Particle Mass Flow > Mass Flow Rate	0.001015 [kg s^-1]
	Boundary Conditions > HC Fuel > Particle Diameter Distribution	(Selected)
	Boundary Conditions > HC Fuel > Particle Diameter Distribution > Option	Discrete Diameter Distribution
	Boundary Conditions > HC Fuel > Particle Diameter Distribution > Diameter List	12, 38, 62, 88 [micron]

Tab	Setting	Value
	Boundary Conditions > HC Fuel > Particle Diameter Distribution > Mass Fraction List	0.18, 0.25, 0.21, 0.36
	Boundary Conditions > HC Fuel > Particle Diameter Distribution > Number Fraction List	0.25, 0.25, 0.25, 0.25
	Boundary Conditions > HC Fuel > Heat Transfer > Option	Static Temperature
	Boundary Conditions > HC Fuel > Heat Transfer > Static Temperature	343 [K]

3. Click **OK**.

30.5.4.2. Air Inlet Boundary

Create the air inlet boundary with mass flow rate and static temperature set consistently with the problem description, as follows:

1. Create a new boundary named `Air Inlet`.
2. Apply the following settings:

Tab	Setting	Value
Basic Settings	Boundary Type	Inlet
	Location	AirInlet
Boundary Details	Mass and Momentum > Option	Mass Flow Rate
	Mass and Momentum > Mass Flow Rate	0.01035 [kg s^-1]
	Flow Direction > Option	Normal to Boundary Condition
	Heat Transfer > Option	Static Temperature
	Heat Transfer > Static Temperature	573 [K]
	Component Details	O2
	Component Details > O2 > Option	Mass Fraction
	Component Details > O2 > Mass Fraction	0.232

3. Click **OK**.

30.5.4.3. Outlet Boundary

Create the outlet boundary with pressure specified, as follows:

1. Create a new boundary named `Outlet`.
2. Apply the following settings:

Tab	Setting	Value
Basic Settings	Boundary Type	Outlet
	Location	Outlet
Boundary Details	Mass and Momentum > Option	Average Static Pressure
	Mass and Momentum > Relative Pressure	0[Pa]
	Mass and Momentum > Pres. Profile Blend	0.05

3. Click **OK**.

30.5.4.4. Coal Gun No-Slip Wall Boundary

Create the `Coal Gun Wall` boundary with a fixed temperature as specified in the problem description, as follows:

1. Create a new boundary named `Coal Gun Wall`.
2. Apply the following settings:

Tab	Setting	Value
Basic Settings	Boundary Type	Wall
	Location	CoalGunWall
Boundary Details	Heat Transfer > Option	Temperature
	Heat Transfer > Fixed Temperature	800 [K]
	Thermal Radiation > Option	Opaque
	Thermal Radiation > Emissivity	0.6 [1 (p. 542)]
	Thermal Radiation > Diffuse Fraction	1

Footnote

1. The wall has an emissivity value of 0.6 since about half of the radiation can travel through the surface and half is reflected and/or absorbed at the surface.

3. Click **OK**.

30.5.4.5. Coal Inlet No-Slip Wall Boundary

Create the `Coal Inlet Wall` boundary with fixed temperature as specified in the problem description, as follows:

1. Create a new boundary named `Coal Inlet Wall`.
2. Apply the following settings:

Tab	Setting	Value
Basic Settings	Boundary Type	Wall

Tab	Setting	Value
	Location	CoalInletInnerWall, CoalInletOuterWall [1 (p. 543)]
Boundary Details	Heat Transfer > Option	Temperature
	Heat Transfer > Fixed Temperature	343 [K]
	Thermal Radiation > Option	Opaque
	Thermal Radiation > Emissivity	0.6
	Thermal Radiation > Diffuse Fraction	1

Footnote

1. Click the *Ellipsis* ... icon to open the **Selection Dialog** dialog box, then select `CoalInletInnerWall` and `CoalInletOuterWall`, holding the **Ctrl** key. Click **OK**.

3. Click **OK**.

30.5.4.6. Air Inlet No-Slip Wall Boundary

Create the `Air Inlet Wall` boundary with fixed temperature as specified in the problem description, as follows:

1. Create a new boundary named `Air Inlet Wall`.
2. Apply the following settings:

Tab	Setting	Value
Basic Settings	Boundary Type	Wall
	Location	AirInletInnerWall, AirInletOuterWall [1 (p. 543)]
Boundary Details	Heat Transfer > Option	Temperature
	Heat Transfer > Fixed Temperature	573 [K]
	Thermal Radiation > Option	Opaque
	Thermal Radiation > Emissivity	0.6
	Thermal Radiation > Diffuse Fraction	1

Footnote

1. Click the *Ellipsis* ... icon to open the **Selection Dialog** dialog box, then select `AirInletInnerWall` and `AirInletOuterWall`, holding the **Ctrl** key. Click **OK**.

3. Click **OK**.

30.5.4.7. Furnace No-Slip Wall Boundary

Create the `Furnace Wall` boundary with a fixed temperature as specified in the problem description, as follows:

1. Create a new boundary named `Furnace Wall`.
2. Apply the following settings:

Tab	Setting	Value
Basic Settings	Boundary Type	Wall
	Location	FurnaceFrontWall, FurnaceOuterWall [1 (p. 544)]
Boundary Details	Heat Transfer > Option	Temperature
	Heat Transfer > Fixed Temperature	1400 [K]
	Thermal Radiation > Option	Opaque
	Thermal Radiation > Emissivity	0.6
	Thermal Radiation > Diffuse Fraction	1

Footnote

1. Click the *Ellipsis* ... icon to open the **Selection Dialog** dialog box, then select `FurnaceFrontWall` and `FurnaceOuterWall`, holding the **Ctrl** key. Click **OK**.

3. Click **OK**.

30.5.4.8. Quarl No-Slip Wall Boundary

1. Create a new boundary named `Quarl Wall`.
2. Apply the following settings:

Tab	Setting	Value
Basic Settings	Boundary Type	Wall
	Location	QuarlWall
Boundary Details	Heat Transfer > Option	Temperature
	Heat Transfer > Fixed Temperature	1200 [K]
	Thermal Radiation > Option	Opaque
	Thermal Radiation > Emissivity	0.6
	Thermal Radiation > Diffuse Fraction	1

3. Click **OK**.

30.5.4.9. Symmetry Plane Boundaries

You will use symmetry plane boundaries on the front and back regions of the cavity.

1. Create a new boundary named `Symmetry Plane 1`.
2. Apply the following settings:

Tab	Setting	Value
Basic Settings	Boundary Type	Symmetry
	Location	PeriodicSide1

3. Click **OK**.
4. Create a new boundary named `Symmetry Plane 2`.
5. Apply the following settings:

Tab	Setting	Value
Basic Settings	Boundary Type	Symmetry
	Location	PeriodicSide2

6. Click **OK**.

30.5.5. Setting Solver Control

1. Click *Solver Control*  .
2. Apply the following settings:

Tab	Setting	Value
Basic Settings	Convergence Control > Max. Iterations	600
	Convergence Control > Fluid Timescale Control > Timescale Control	Physical Timescale
	Convergence Control > Fluid Timescale Control > Physical Timescale	0.005 [s] [1 (p. 546)]
Particle Control	Particle Coupling Control > First Iteration for Particle Calculation	(Selected)
	Particle Coupling Control > First Iteration for Particle Calculation > First Iteration	25 [2 (p. 546)]
	Particle Coupling Control > Iteration Frequency	(Selected)
	Particle Coupling Control > Iteration Frequency > Iteration Frequency	10 [3 (p. 546)]
	Particle Under Relaxation Factors	(Selected)
	Particle Under Relaxation Factors > Vel. Under Relaxation	0.75
	Particle Under Relaxation Factors > Energy	0.75

Tab	Setting	Value
	Particle Under Relaxation Factors > Mass	0.75
	Particle Ignition	(Selected)
	Particle Ignition > Ignition Temperature	1000 [K]
	Particle Source Smoothing	(Selected)
	Particle Source Smoothing > Option	Smooth
Advanced Options	Thermal Radiation Control	(Selected)
	Thermal Radiation Control > Coarsening Control	(Selected)
	Thermal Radiation Control > Coarsening Control > Target Coarsening Rate	(Selected)
	Thermal Radiation Control > Coarsening Control > Target Coarsening Rate > Rate	16 [4 (p. 546)]

Footnotes

1. Based on the air inlet speed and the size of the combustor.
2. The First Iteration parameter sets the coefficient-loop iteration number at which particles are first tracked; it allows the continuous-phase flow to develop before tracking droplets through the flow. Experience has shown that the value usually has to be increased to 25 from the default of 10.
3. The Iteration Frequency parameter is the frequency at which particles are injected into the flow after the First Iteration for Particle Calculation iteration number. The iteration frequency allows the continuous phase to settle down between injections because it is affected by sources of momentum, heat, and mass from the droplet phase. Experience has shown that the value usually has to be increased to 10 from the default of 5.
4. The Target Coarsening Rate parameter controls the size of the radiation element required for calculating Thermal Radiation. Decreasing the size of the element to 16, from the default 64, increases the accuracy of the solution obtained, while increasing the computing time required for the calculations.

3. Click **OK**.

30.5.6. Writing the CFX-Solver Input (.def) File

1. Click *Define Run*  .
2. Apply the following settings:

Setting	Value
File name	CoalCombustion_nonox.def

3. Click **Save**.

   CFX-Solver Manager automatically starts and, on the **Define Run** dialog box, the **Solver Input File** is set.

4. Quit CFX-Pre, saving the simulation (`.cfx`) file.

30.5.7. Obtaining a Solution for the Simulation using CFX-Solver Manager

When CFX-Pre has shut down and the CFX-Solver Manager has started, obtain a solution to the CFD problem by following the instructions below:

1. Ensure that the **Define Run** dialog box is displayed.

   Solver Input File should be set to `CoalCombustion_nonox.def`.

2. Click **Start Run**.

   CFX-Solver runs and attempts to obtain a solution. This can take a long time depending on your system. Eventually a dialog box is displayed.

3. At the end of the run, on the completion message that appears, select **Post-Process Results**.

4. If using Standalone mode, select **Shut down CFX-Solver Manager**.

5. Click **OK**.

30.5.8. Viewing the Results in CFD-Post

In this section, you will make plots showing the variation of temperature, water mass fraction, and radiation intensity on the `Symmetry Plane 1` boundary. You will also color the particle tracks, which were produced by the solver and included in the results file, by temperature and by ash mass fraction. The particle tracks help to illustrate the mean flow behavior in the coal furnace.

30.5.8.1. Displaying the Temperature on a Symmetry Plane

1. Right-click a blank area in the viewer and select **Predefined Camera** > **Isometric View (Z up)**.

   This orients the geometry with the inlets at the top, as shown at the beginning of this tutorial.

2. Edit `Cases` > `CoalCombustion_nonox_001` > `Furnace` > `Symmetry Plane 1`.

3. Apply the following settings:

Tab	Setting	Value
Color	Mode	Variable
	Variable	Temperature
Render	Show Faces	(Selected)
	Lighting	(Cleared) [1 (p. 547)]

Footnote

1. Turning off the lighting makes the colors accurate in the 3D view, but can make it more difficult to perceive depth. As an alternative to turning off the lighting, you can try rotating the view to a different position.

4. Click **Apply**.

   As expected for a non-swirling case, the flame appears a significant distance away from the burner. The flame is likely unstable, as evidenced by the rate of solver convergence; the next simulation in this tutorial involves swirl, which tends to stabilize the flame, and has much faster solver convergence.

30.5.8.2. Displaying the Water Mass Fraction

Change the variable used for coloring the plot to `H20.Mass Fraction` and click **Apply**.

From the plot it can be seen that water is produced a significant distance away from the burner, as was the flame in the previous plot. As expected, the mass fraction of water is high where the temperature is high.

30.5.8.3. Displaying the Radiation Intensity

1. Change the variable used for coloring the plot to `Radiation Intensity` and click **Apply**.

   This plot is directly related to the temperature plot. This result is consistent with radiation being proportional to temperature to the fourth power.

2. When you are finished, right-click `Symmetry Plane 1` in the **Outline** tree view and select **Hide**.

30.5.8.4. Displaying the Temperature of the Fuel Particles

Color the existing particle tracks for the solid particles by temperature:

1. Edit `Cases` > `CoalCombustion_nonox_001` > `Res PT for HC Fuel`.
2. Apply the following settings:

Tab	Setting	Value
Color	Mode	Variable
	Variable	HC Fuel.Temperature

3. Click **Apply**.

   Observing the particle tracks, you can see that coal enters the chamber at a temperature of around 343 K. The temperature of the coal, as it moves away from the inlet, rises as it reacts with the air entering from the inlet. The general location where the temperature of the coal increases rapidly is close to the location where the flame appears to be according to the plots created earlier. Downstream of this location, the temperature of the coal particles begins to drop.

30.5.8.5. Displaying the Ash Mass Fraction using Particle Tracking

1. Change the plot of the particle tracks so that they are colored by `HC Fuel.Ash.Mass Fraction`.

   The ashes form in the flame region, as expected.

2. Quit CFD-Post, saving the state (`.cst`) file at your discretion.

30.6. Coal Combustion Simulation with Swirl

You will now create a simulation where swirl burners are used and where there is no release of nitrogen oxide during the process. Swirl burners inject a fuel axially into the combustion chamber surrounded by an

annular flow of oxidant (normally air) which has, upon injection, some tangential momentum. This rotational component, together with the usually divergent geometry of the burner mouth, cause two important effects:

- They promote intense mixing between fuel and air, which is important for an efficient and stable combustion, and low emissions.
- They originate a recirculation region, just at the burner mouth, which traps hot combustion products and acts as a permanent ignition source, hence promoting the stability of the flame.

If you want to set up the simulation automatically using a tutorial session file, run `CoalCombustion_nonox_swirl.pre`. For details, see *Playing a Tutorial Session File* (p. 4). Then proceed to *Obtaining a Solution for the Simulation with Swirl using CFX-Solver Manager* (p. 551).

If you want to set up the simulation manually, proceed to the following section.

30.6.1. Opening an Existing Simulation

1. If CFX-Pre is not already running, start it.
2. Select **File** > **Open Case**.
3. From your working directory, select `CoalCombustion_nonox.cfx` and click **Open**.
4. Select **File** > **Save Case As**.
5. Set **File name** to `CoalCombustion_nonox_swirl.cfx`.
6. Click **Save**.

30.6.2. Editing the Boundary Conditions

To add swirl to the flow, you will edit the `Air Inlet` boundary to change the flow direction specification from `Normal to Boundary Condition` to `Cylindrical Components`. You will also edit the `Outlet` boundary to change the `Pressure Profile Blend` setting from 0.05 to 0; the reason for this change is explained later. You will also delete the two symmetry plane boundary conditions and replace them with a periodic domain interface.

30.6.2.1. Air Inlet Boundary

1. Edit `Simulation` > `Flow Analysis 1` > `Furnace` > `Air Inlet`.
2. Apply the following settings:

Tab	Setting	Value
Boundary Details	Flow Direction > Option	Cylindrical Components
	Flow Direction > Axial Component	0.88
	Flow Direction > Radial Component	0
	Flow Direction > Theta Component	1
	Axis Definition > Rotational Axis	Global Z

3. Click **OK**.

30.6.2.2. Outlet Boundary

The average pressure boundary condition leaves the pressure profile unspecified while constraining the average pressure to the specified value. In some situations, leaving the profile fully unspecified is too weak and convergence difficulties may result. The 'Pressure Profile Blend' feature works around this by blending between a unspecified pressure profile and a fully specified pressure profile. By default, the pressure profile blend is 5%. For swirling flow, however, imposing any amount of a uniform pressure profile is inconsistent with the radial pressure profile which should naturally develop in response to the fluid rotation, and the pressure profile blend must be set to 0.

1. Edit `Simulation` > `Flow Analysis 1` > `Furnace` > `Outlet`.
2. Apply the following settings:

Tab	Setting	Value
Boundary Details	Mass and Momentum > Option	Average Static Pressure
	Mass and Momentum > Pres. Profile Blend	0

3. Click **OK**.

30.6.2.3. Deleting the Symmetry Plane Boundaries

1. In the **Outline** tree view, right-click `Simulation` > `Flow Analysis 1` > `Furnace` > `Symmetry Plane 1`, then select **Delete**.
2. Repeat step 1 to delete `Symmetry Plane 2`.

30.6.3. Creating a Domain Interface

You will insert a domain interface to connect the `Periodic Side 1` and `Periodic Side 2` regions.

1. Create a domain interface named `Periodic`.
2. Apply the following settings:

Tab	Setting	Value
Basic Settings	Interface Type	Fluid Fluid
	Interface Side 1 > Region List	PeriodicSide1
	Interface Side 2 > Region List	PeriodicSide2
	Interface Models > Option	Rotational Periodicity

3. Click **OK**.

30.6.4. Writing the CFX-Solver Input (.def) File

1. Click *Define Run*  .
2. Apply the following settings:

Setting	Value
File name	CoalCombustion_nonox_swirl.def

3. Click **Save**.

   CFX-Solver Manager automatically starts and, on the **Define Run** dialog box, the **Solver Input File** is set.

4. Quit CFX-Pre, saving the simulation (`.cfx`) file.

30.6.5. Obtaining a Solution for the Simulation with Swirl using CFX-Solver Manager

When CFX-Pre has shut down and the CFX-Solver Manager has started, obtain a solution to the CFD problem by following the instructions below.

1. Ensure that the **Define Run** dialog box is displayed.

   Solver Input File should be set to `CoalCombustion_nonox_swirl.def`.

2. Click **Start Run**.

   CFX-Solver runs and attempts to obtain a solution. This can take a long time depending on your system. Eventually a dialog box is displayed.

3. At the end of the run, on the completion message that appears, select **Post-Process Results**.
4. If using Standalone mode, select **Shut down CFX-Solver Manager**.
5. Click **OK**.

30.6.6. Viewing the Results for Simulation with Swirl in CFD-Post

In this section, you will make plots showing the variation of temperature, water mass fraction, and radiation intensity on the `Periodic Side 1` boundary. You will also color the existing particle tracks by temperature and by ash mass fraction.

30.6.6.1. Displaying the Temperature on a Periodic Interface

1. Right-click a blank area in the viewer and select **Predefined Camera** > **Isometric View (Z up)**.
2. Edit `Cases` > `CoalCombustion_nonox_swirl_001` > `Furnace` > `Periodic Side 1`.
3. Apply the following settings:

Tab	Setting	Value
Color	Mode	Variable
	Variable	Temperature
Render	Show Faces	(Selected)
	Lighting	(Cleared)

4. Click **Apply**.

As expected, the flame appears much closer to the burner than in the previous simulation which had no swirl. This is due to the fact that the swirl component applied to the air from `Air Inlet` tends to entrain coal particles and keep them near the burner for longer, thus helping them to burn.

30.6.6.2. Displaying the Water Mass Fraction

Change the variable used for coloring the plot to `H2O.Mass Fraction` and click **Apply**.

Similar to the no-swirl case, the mass fraction of water with swirl is directly proportional to the temperature of the furnace.

30.6.6.3. Displaying the Radiation Intensity

1. Change the variable used for coloring the plot to `Radiation Intensity` and click **Apply**.
2. When you are finished, right-click `Periodic Side 1` in the **Outline** tree view and select **Hide**.

30.6.6.4. Displaying the Temperature using Particle Tracking

1. Edit `Cases` > `CoalCombustion_nonox_swirl_001` > `Res PT for HC Fuel`.
2. Apply the following settings:

Tab	Setting	Value
Color	Mode	Variable
	Variable	HC Fuel.Temperature

3. Click **Apply**.

30.6.6.5. Displaying the Ash Mass Fraction using Particle Tracking

1. Change the plot of the particle tracks so that they are colored by `HC Fuel.Ash.Mass Fraction`.
2. Quit CFD-Post, saving the state (`.cst`) file at your discretion.

30.7. Coal Combustion Simulation with Swirl and Nitrogen Oxide

You will now create a simulation that involves both swirl and the release of nitrogen oxide. The CCL file that was previously imported contains the nitrogen oxide material,`NO`, and reactions,`Prompt NO Fuel Gas PDF` and `Thermal NO PDF`, required for this combustion simulation. Nitrogen oxide is calculated as a postprocessing step in the solver.

If you want to set up the simulation automatically using a tutorial session file, run `CoalCombustion_nox-cpp_swirl.pre`. For details, see *Playing a Tutorial Session File* (p. 4). Then proceed to *Obtaining a Solution for the Simulation with Swirl and Nitrogen Oxide using CFX-Solver Manager* (p. 554).

If you want to set up the simulation manually, proceed to the following section.

30.7.1. Opening an Existing Simulation

1. If CFX-Pre is not already running, start it.
2. Select **File** > **Open Case**.
3. From your working directory, select `CoalCombustion_nonox_swirl.cfx` and click **Open**.

4. Select **File** > **Save Case As**.
5. Set **File name** to `CoalCombustion_noxcpp_swirl.cfx`.
6. Click **Save**.

30.7.2. Editing the Domain

In this section, you will edit the `Furnace` domain by adding the new material `NO` to the materials list. CFX-Solver requires that you specify enough information for the mass fraction of `NO` at each of the system inlets. In this case, set the `NO` mass fraction at the air and coal inlets to zero.

1. Edit `Simulation` > `Flow Analysis 1` > `Furnace`.
2. Apply the following settings:

Tab	Setting	Value
Fluid Specific Models	Fluid	Gas Mixture
	Fluid > Gas Mixture > Combustion > Chemistry Post Processing	(Selected)
	Fluid > Gas Mixture > Combustion > Chemistry Post Processing > Materials List	NO
	Fluid > Gas Mixture > Combustion > Chemistry Post Processing > Reactions List	Prompt NO Fuel Gas PDF,Thermal NO PDF

   These settings enable the combustion simulation with nitrogen oxide (NO) as a postprocessing step in the solver. The NO reactions are defined in the same way as any participating reaction but the simulation of the NO reactions is performed after the combustion simulation of the air and coal. With this one-way simulation, the NO will have no effect on the combustion simulation of the air and coal.
3. Click **OK**.
4. Edit `Simulation` > `Flow Analysis 1` > `Furnace` > `Air Inlet`.
5. Apply the following settings:

Tab	Setting	Value
Boundary Details	Component Details	NO
	Component Details > NO > Mass Fraction	0.0

6. Click **OK**.
7. Edit `Simulation` > `Flow Analysis 1` > `Furnace` > `Coal Inlet`.
8. Apply the following settings:

Tab	Setting	Value
Boundary Details	Component Details	NO
	Component Details > NO > Mass Fraction	0.0

9. Click **OK**.

30.7.3. Writing the CFX-Solver Input (.def) File

1. Click *Define Run*  .
2. Apply the following settings:

Setting	Value
File name	CoalCombustion_noxcpp_swirl.def

3. Click **Save**.
4. Quit CFX-Pre, saving the simulation as `CoalCombustion_noxcpp_swirl.cfx`.

30.7.4. Obtaining a Solution for the Simulation with Swirl and Nitrogen Oxide using CFX-Solver Manager

When CFX-Pre has shut down and the CFX-Solver Manager has started, obtain a solution to the CFD problem by following the instructions below.

1. Ensure that the **Define Run** dialog box is displayed.

   Solver Input File should be set to `CoalCombustion_noxcpp_swirl.def`.
2. Select **Initial Values Specification**.
3. Select `CoalCombustion_nonox_swirl_001.res` for the initial values file using the *Browse*  tool.

   The fluid solution from the previous case has not changed for this simulation. Loading the results from the previous case as an initial guess eliminates the need for the solver to solve for the fluids solutions again.
4. Click **Start Run**.

   CFX-Solver runs and attempts to obtain a solution. This can take a long time depending on your system. Eventually a dialog box is displayed stating that the run has finished.
5. At the end of the run, on the completion message that appears, select **Post-Process Results**.
6. If using Standalone mode, select **Shut down CFX-Solver Manager**.
7. Click **OK**.

30.7.5. Viewing the Results for the Simulation with Swirl and Nitrogen Oxide in CFD-Post

In this section, you will make a plot on the `Periodic Side 1` region showing the variation of concentration of nitrogen oxide through the coal furnace.

1. Right-click a blank area in the viewer and select **Predefined Camera** > **Isometric View (Z up)**.
2. Edit `Cases` > `CoalCombustion_noxcpp_swirl_001` > `Furnace` > `Periodic Side 1`.
3. Apply the following settings:

Tab	Setting	Value
Color	Mode	Variable
	Variable	NO.Mass Fraction
Render	Show Faces	(Selected)
	Lighting	(Cleared)

4. Click **Apply**.

   You can see that NO is produced in the high-temperature region near the inlet. Further downstream, the mass fraction of NO is more uniform.

5. Quit CFD-Post, saving the state (`.cst`) file at your discretion.

Chapter 31: Steam Jet

This tutorial includes:

31.1. Tutorial Features

In this tutorial you will learn about:

- Importing a CCL file in CFX-Pre.
- High speed multi-component, multiphase flow with interphase mass transfer.
- Model customization using CEL.
- Handling mass sources based on species transfer.
- Source linearization.

Component	Feature	Details
CFX-Pre	User Mode	General Mode
	Domain Type	Single Domain
	Analysis Type	Steady State
	Fluid Type	Continuous Fluid
		Dispersed Fluid
	CCL File	Import
	Boundary Conditions	Inlet Boundary
		Opening Boundary
		Outlet Boundary
		Steam Jet Default
		Symmetry Boundary
	Timestep	Physical Timescale
CFD-Post	Plots	Default Locators
		Line Locator
	Other	Chart Creation

31.2. Overview of the Problem to Solve

This tutorial investigates the simulation of a high-speed wet steam jet into air. Such a simulation might be produced by a leaking steam, or high-pressure hot water pipe, just down stream of the actual leak point. The air is cold and dry, causing the steam to condense further as it mixes with the air. This tutorial is based on a two-fluid model and comprises of the following components: a pure liquid, a disperse phase fluid representing the water and a two-component gas, and a continuous phase fluid representing the steam and air. Mass transfer occurs between two fluids as the water condenses or evaporates. This is simulated explicitly using mass sources and sinks in the two fluids. The mass transfer itself is modeled as a return to equilibrium based on the difference between the actual molar concentration of steam in air to the saturation value. The rate of mass transfer is modeled using a very simple Sherwood number-based mass diffusion at the surface of liquid drops. The geometry is two dimensional and cylindrically symmetric with the far field modeled using an outlet normal to the symmetry axis down stream and an opening in all other directions.

The steam jet has an inlet at the end of an injection pipe, where the gas and liquid enter the system at a normal speed of 341 m s^-1. Symmetry boundaries are used on two sides of the domain because a thin section of the geometry is modeled and there is no swirl. An opening boundary is used around the outside edges of the domain; the opening condition prescribes a flow direction normal to the boundary in order to provide sufficient constraints on the solution.

Some of the relevant parameters of this problem are:

- Static temperature of the injected gas and liquid = 373 K
- Average static pressure around the domain = 0 Pa
- Temperature around the domain = 25 °C

To set up this simulation, you will first import the mesh and CCL files that contain the required expressions and Additional Variable definitions. You will then define:

- the required materials
- a domain that involves both gas and liquid water
- subdomains that account for gas-to-liquid and liquid-to-gas phase changes
- boundary conditions

31.3. Before You Begin

It is strongly recommended that you complete the previous tutorials before trying this one. However, if this is the first tutorial you are working with, it is important to review the following topics before beginning:

- *Preparing a Working Directory and Starting ANSYS CFX in Standalone Mode* (p. 1)
- *Running ANSYS CFX Tutorials in ANSYS Workbench* (p. 2)
- *Changing the Display Colors* (p. 5)
- *Playing a Tutorial Session File* (p. 4)

31.4. Starting CFX-Pre

1. Prepare the working directory using the following files in the `examples` directory:
   - `SteamJet.pre`
   - `steam_jet.gtm`

- `steam_jet_expressions.ccl`
- `steam_jet_additional_variables.ccl`

2. Set the working directory and start CFX-Pre.

   For details, see *Preparing a Working Directory and Starting ANSYS CFX in Standalone Mode* (p. 1)

31.5. Defining a Case in CFX-Pre

If you want to set up the case automatically using a tutorial session file, run `SteamJet.pre`. For details, see *Playing a Tutorial Session File* (p. 4). Then proceed to *Obtaining a Solution Using CFX-Solver Manager* (p. 574).

If you want to set up the case manually, proceed to the following steps:

This section describes the step-by-step definition of the flow physics in CFX-Pre for a steady-state simulation.

1. In CFX-Pre, select **File** > **New Case**.
2. Select **General** and click **OK**.
3. Edit `Case Options` > `General` in the **Outline** tree view and ensure that **Automatic Default Domain** and **Automatic Default Interfaces** are turned off.
4. Select **File** > **Save Case As**.
5. Set **File name** to `SteamJet.cfx`.
6. Click **Save**.

31.5.1. Importing the Mesh

1. Select **File** > **Import** > **Mesh**.
2. In the **Import Mesh** dialog box, set **Files of type** to `CFX Mesh (*gtm *cfx)` and select `steam_jet.gtm` from your working directory.
3. Click **Open**.

31.5.2. Importing the Steam Jet CCL

ANSYS CFX Command Language (CCL) consists of commands used to carry out actions in CFX-Pre, CFX-Solver Manager, and CFD-Post. Expressions and Additional Variables required for the steam jet simulation will be imported from CCL files. This section outlines the steps to analyze `steam_jet_expressions.ccl` and `steam_jet_additional_variables.ccl`, and then import them into the simulation.

Note

The physics for a case can be saved to a CCL (CFX Command Language) file at any time by selecting **File** > **Export** > **CCL**.

1. Select CCL files from your working directory, and open them one at a time with a text editor and take the time to look at the information they contain. For details on setting up the working directory, see *Starting CFX-Pre* (p. 558). The information contained in the CCL files is outlined below:

   - The CCL file `steam_jet_expressions.ccl` creates expressions required for setting up the following data:

  - Liquid/gas interface
  - Interphase diffusive transport coefficient
  - Heats of vaporization
  - Liquid-gas mass transfer for water
  - Continuity linearization with respect to P,
  - Local false step linearization of the IPMT.
- The CCL file `steam_jet_additional_variables.ccl` creates the following Additional Variables:
  - Pressure linearization coefficient `PCoef`
  - Water IPMT flux liquid to gas `WaFluxLG`
  - Water IPMT flux gas to liquid `WaFluxGL`
  - Local IPMT false timestep `FalseDt`
  - Saturation temperature for post `SatTemp`
  - Saturation pressure for post `SatPres`
  - Latent heat at saturation for post `SatLHeat`

2. Select **File** > **Import** > **CCL**

   The **Import CCL** dialog box appears.
3. Under **Import Method**, select **Append**. This option appends the changes to the existing case.

   Note

   The **Replace** option is useful if you have defined the physics and want to update or replace the existing physics using the newly imported CCL.

4. From your working directory, select `steam_jet_expressions.ccl`.
5. Click **Open**. The CCL is now loaded as indicated by the status bar in the bottom right corner of the window. After a short pause, the CCL and the **Outline** tree view will be updated.
6. Repeat steps 2 to 5 to import `steam_jet_additional_variables.ccl`.
7. In the **Outline** tree view, expand the `Additional Variables` and `Expressions` branches under `Simulation` > `Expressions, Functions and Variables` to confirm that new objects have been added after importing the CCL files.

31.5.3. Creating a Steady State Analysis

The characteristics of this case do not change as a function of time, and therefore a steady state analysis is appropriate.

1. Click *Analysis Type*  .
2. Apply the following settings:

Tab	Setting	Value
Basic Settings	External Solver Coupling > Option	None
	Analysis Type > Option	Steady State

3. Click **OK**.

31.5.4. Creating and Loading Materials

In addition to providing template fluids, CFX allows you to create custom fluids for use in all your CFX models. A custom fluid may be defined as a pure substance, but may also be defined as a mixture, consisting of a number of transported fluid components. This type of fluid model is useful for applications involving mixtures, reactions, and combustion.

In order to define custom fluids, CFX-Pre provides the **Material** details view. This tool allows you to define your own fluids as pure substances, fixed composition mixtures or variable composition mixtures using a range of template property sets defined for common materials.

The steam jet application requires two mixtures made up from three separate materials (or components). You are first going to load the materials that take part in the process (`Steam3v` and `Steam3l`). The `Air Ideal Gas` material is already loaded. Finally, you will create a variable composition mixture as well as a fixed composition mixture consisting of these three materials. In a variable composition mixture, the proportion of each component will change throughout the simulation; while in a fixed composition mixture, the proportion of each component is fixed.

31.5.4.1. Loading the Steam3l, Steam3v, and Steam3vl Materials

Load the materials `Steam3l`, `Steam3v`, and `Steam3vl` from the CFX-Pre Materials Library.

1. In the **Outline** tree view, right-click `Simulation` > `Materials` and select **Import Library Data**.

   The **Select Library Data to Import** dialog box appears.

2. Click the browse button  next to **File to Import**.

   The **Import CCL** dialog box appears.

3. Under **Import Method**, select **Append**. This options appends the CCL changes to the existing case.
4. Select `MATERIALS-iapws.ccl` from the `etc/materials-extra` directory and click **Open**.
5. In the **Select Library Data to Import** dialog box, expand `Wet Steam` and select `Steam3vl`.
6. Click **OK**.
7. In the **Outline** tree view, expand `Simulation` > `Materials` to confirm that `Steam3l`, `Steam3v`, and `Steam3vl` have been added to the list.

31.5.4.2. Creating the Gas Mixture Material

Create a new material named `Gas mixture` that will be composed of `Air Ideal Gas` and `Steam3v`. This material will be injected into the gas inlet during the steady state simulation.

1. Create a new material named `Gas mixture`.
2. Apply the following settings:

Tab	Setting	Value
Basic Settings	Option	Variable Composition Mixture
	Materials List	Air Ideal Gas, Steam3v [1 (p. 562)]

Footnote

1. Click the *Ellipsis* icon ... to open the **Materials List** dialog box, then select multiple items by holding the **Ctrl** key. Click **OK**.

3. Click **OK**.

31.5.4.3. Creating the Liquid Mixture Material

Create a new material named `Liquid mixture` that will be composed of `Steam3l`.

1. Create a new material named `Liquid mixture`.
2. Apply the following settings:

Tab	Setting	Value
Basic Settings	Option	Fixed Composition Mixture
	Materials List	Steam3l
	Child Materials > Steam3l > Mass Fraction	1.0

3. Click **OK**.

31.5.5. Creating the Domain

This section outlines the steps to create a new domain `Steam Jet`.

1. Select **Insert** > **Domain** from the menu bar, or click *Domain* .
2. In the **Insert Domain** dialog box, set the name to `Steam Jet` and click **OK**.
3. On the **Basic Settings** tab, apply the following settings under **Location and Type**:

Setting	Value
Location	B26
Domain Type	Fluid Domain
Coordinate Frame	Coord 0

4. On the **Basic Settings** tab, delete any existing items under **Fluid and Particle Definitions** by selecting them and clicking *Remove selected item* .

5. Under **Fluid and Particle Definitions**, create two fluid definitions named `Gas` and `Liquid` by using the *Add new item*  icon.

   The new fluids named `Gas` and `Liquid` appear under **Fluid and Particle Definitions**.

6. On the **Basic Settings** tab, apply the following settings under **Fluid and Particle Definitions**:

Setting	Value
(List Box)	Gas
Gas > Material	Gas mixture
Gas > Morphology > Option	Continuous Fluid
(List Box)	Liquid
Liquid > Material	Liquid mixture
Liquid > Morphology > Option	Dispersed Fluid
Liquid > Morphology > Mean Diameter	liqLength [1 (p. 563)]

Footnote

1. Click the *Enter Expression* icon  to specify the CEL expression.

7. On the **Fluid Models** tab, apply the following settings:

Setting	Value
Heat Transfer > Option	Fluid Dependent
Turbulence > Option	Fluid Dependent
Combustion > Option	None
Thermal Radiation > Option	None

8. On the **Fluid Models** tab under **Additional Variable Models** > **Additional Variable**, select `FalseDt` and select the **FalseDt** check box.
9. Make sure that **Additional Variable Models** > **Additional Variable** > **FalseDt** > **Option** is set to `Fluid Dependent`.
10. Repeat the previous two steps for the rest of the Additional Variables (PCoef, SatLheat, SatPres, SatTemp, WaFluxGL, WaFluxLG).
11. On the **Fluid Specific Models** tab, select `Gas` in the list box, then apply the following settings:

Setting	Value
Heat Transfer Model > Option	Total Energy
Turbulence > Option	k-Epsilon
Turbulence > Wall Function > Wall Function	Scalable
Component Models > Component	Air Ideal Gas
Component Models > Component > Air Ideal Gas > Option	Constraint

Setting	Value
Component Models > Component	Steam3v
Component Models > Component > Steam3v > Option	Transport Equation
Component Models > Component > Steam3v > Kinematic Diffusivity	(Selected)
Component Models > Component > Steam3v > Kinematic Diffusivity > Kinematic Diffusivity	KinDiff [1 (p. 564)]
Additional Variable Models	PCoef
Additional Variable Models > PCoef	(Selected)
Additional Variable Models > PCoef > Add. Var. Value	dFLUXwadp [1 (p. 564)]

Footnote

1. Click the *Enter Expression* icon  to specify the CEL expression.

12. On the **Fluid Specific Models** tab, select `Liquid` in the list box, then apply the following settings:

Setting	Value
Heat Transfer Model > Option	Total Energy
Turbulence > Option	Dispersed Phase Zero Equation

13. Under **Additional Variable Models** (for Liquid), select `FalseDt` in the list box, then apply the following settings:

Setting	Value
FalseDt	(Selected)
FalseDt > Add. Var. Value	DtFalseMf [1 (p. 564)]

Footnote

1. Click the *Enter Expression* icon  to specify the CEL expression.

14. Repeat the previous step for the rest of the Additional Variables (PCoef, SatLheat, SatPres, SatTemp, WaFluxGL, WaFluxLG) using the following values:

Additional Variable	Expression
PCoef	dFLUXwadp [1 (p. 565)]
SatLheat	HtVapwa [1 (p. 565)]
SatPres	VpWat [1 (p. 565)]

Additional Variable	Expression
SatTemp	SatT [1 (p. 565)]
WaFluxGL	FLUXwa1 [1 (p. 565)]
WaFluxLG	FLUXwa2 [1 (p. 565)]

Footnote

1. Click the *Enter Expression* icon  to specify the CEL expression.

15. On the **Fluid Pair Models** tab, select `Gas | Liquid` in the list box, then apply the following settings:

Setting	Value
Surface Tension Coefficient	(Selected)
Surface Tension Coefficient > Surf. Tension Coeff.	srfTenCoef [1 (p. 565)]
Interphase Transfer > Option	Particle Model
Momentum Transfer > Drag Force > Option	Schiller Naumann
Turbulence Transfer > Option	None
Mass Transfer > Option	None
Heat Transfer > Option	Ranz Marshall

Footnote

1. Click the *Enter Expression* icon  to specify the CEL expression.

16. Click **OK**.

31.5.6. Creating Subdomains

To provide the correct modeling for the steam jet you need to define mass fraction sources for the fluid components `steam3v` and `steam3l`. To do this, you need to create a subdomain where the relevant sources can be specified. In this case, sources need to be provided within the entire domain of the steam jet since the reaction occurs throughout the domain.

31.5.6.1. Gas to Liquid Source Subdomain

This section outlines the steps to create a new subdomain `GastoLiq`.

1. Create a subdomain named `GastoLiq`.
2. On the **Basic Settings** tab, apply the following settings:

Setting	Value
Location	B26

Setting	Value
Coordinate Frame	Coord 0

3. On the **Fluid Sources** tab, select Gas in the list box, then select the **Gas** check box and apply the following settings:

Setting	Value
Equation Sources	Continuity
Equation Sources > Continuity	(Selected)
Equation Sources > Continuity > Option	Fluid Mass Source
Equation Sources > Continuity > Source	-Liquid.WaFluxGL [1 (p. 567)]
Equation Sources > Continuity > MCF/Energy Sink Option	(Selected)
Equation Sources > Continuity > MCF/Energy Sink Option > Sink Option	Spec. Mass Frac. and Loc. Temp.
Equation Sources > Continuity > Mass Source Volume Fraction Coefficient	(Selected)
Equation Sources > Continuity > Mass Source Volume Fraction Coefficient > Volume Frac. Coeff.	-Gas.density/DtFalseMf [1 (p. 567)]
Equation Sources > Continuity > Variables > Steam3v.mf > Option	Value
Equation Sources > Continuity > Variables > Steam3v.mf > Value	1
Equation Sources > Continuity > Variables > Temperature > Option	Value
Equation Sources > Continuity > Variables > Temperature > Value	Gas.T [1 (p. 567)]
Equation Sources > Continuity > Variables > Turbulence Eddy Dissipation > Option	Value
Equation Sources > Continuity > Variables > Turbulence Eddy Dissipation > Value	Gas.ed [1 (p. 567)]
Equation Sources > Continuity > Variables > Turbulence Kinetic Energy > Option	Value
Equation Sources > Continuity > Variables > Turbulence Kinetic Energy > Value	Gas.ke [1 (p. 567)]
Equation Sources > Continuity > Variables > Velocity > Option	Cartesian Vector Components
Equation Sources > Continuity > Variables > Velocity > U	Gas.Velocity u [1 (p. 567)]
Equation Sources > Continuity > Variables > Velocity > V	Gas.Velocity v [1 (p. 567)]
Equation Sources > Continuity > Variables > Velocity > W	Gas.Velocity w [1 (p. 567)]
Equation Sources	Steam3v.mf
Equation Sources > Steam3v.mf	(Selected)
Equation Sources > Steam3v.mf > Option	Source

Setting	Value
Equation Sources > Steam3v.mf > Source	0 [kg m^-3 s^-1]
Equation Sources > Steam3v.mf > Source Coefficient	(Selected)
Equation Sources > Steam3v.mf > Source Coefficient > Source Coefficient	dFLwadYG [1 (p. 567)][2 (p. 567)]

Footnotes

1. Click the *Enter Expression* icon  to specify the CEL expression.
2. This source coefficient is required only for the mass transfer rates between gas and liquid phases. The source is set to 0 [kg m^3 s^-1] because there is no external source and thus no additional mass is transferred into the system.

4. On the **Fluid Sources** tab, select `Liquid` in the list box, then select the `Liquid` check box and apply the following settings:

Setting	Value
Equation Sources	Continuity
Equation Sources > Continuity	(Selected)
Equation Sources > Continuity > Option	Fluid Mass Source
Equation Sources > Continuity > Source	Liquid.WaFluxGL [1 (p. 568)]
Equation Sources > Continuity > Mass Source Volume Fraction Coefficient	(Selected)
Equation Sources > Continuity > Mass Source Volume Fraction Coefficient > Volume Frac. Coeff.	-Liquid.density/DtFalseMf [1 (p. 568)]
Equation Sources > Continuity > Variables > Temperature > Option	Value
Equation Sources > Continuity > Variables > Temperature > Value	Gas.T [1 (p. 568)]
Equation Sources > Continuity > Variables > Velocity > Option	Cartesian Vector Components
Equation Sources > Continuity > Variables > Velocity > U	Gas.Velocity u [1 (p. 568)]
Equation Sources > Continuity > Variables > Velocity > V	Gas.Velocity v [1 (p. 568)]
Equation Sources > Continuity > Variables > Velocity > W	Gas.Velocity w [1 (p. 568)]
Equation Sources	Energy
Equation Sources > Energy	(Selected)
Equation Sources > Energy > Option	Source
Equation Sources > Energy > Source	Liquid.WaFluxGL*HtVapwa [1 (p. 568)]
Equation Sources > Energy > Source Coefficient	(Selected)
Equation Sources > Energy > Source Coefficient > Source Coefficient	-Liquid.vf*Liquid.density*Liquid.Cp/DtFalseMf [1 (p. 568)]

Footnote

1. Click the *Enter Expression* icon  to specify the CEL expression.

5. Click **OK**.

31.5.6.2. Liquid to Gas Source Subdomain

This section outlines the steps to create a new subdomain `LiqtoGas`.

1. Create a new subdomain named `LiqtoGas`.
2. On the **Basic Settings** tab, apply the following settings:

Setting	Value
Location	B26
Coordinate Frame	Coord 0

3. On the **Fluid Sources** tab, select `Gas` in the list box, then select the **Gas** check box and apply the following settings:

Setting	Value
Equation Sources	Continuity
Equation Sources > Continuity	(Selected)
Equation Sources > Continuity > Option	Fluid Mass Source
Equation Sources > Continuity > Source	Liquid.WaFluxLG [1 (p. 569)]
Equation Sources > Continuity > Variables > Steam3v.mf > Option	Value
Equation Sources > Continuity > Variables > Steam3v.mf > Value	1
Equation Sources > Continuity > Variables > Temperature > Option	Value
Equation Sources > Continuity > Variables > Temperature > Value	SatT [1 (p. 569)]
Equation Sources > Continuity > Variables > Turbulence Eddy Dissipation > Option	Value
Equation Sources > Continuity > Variables > Turbulence Eddy Dissipation > Value	Gas.ed [1 (p. 569)]
Equation Sources > Continuity > Variables > Turbulence Kinetic Energy > Option	Value
Equation Sources > Continuity > Variables > Turbulence Kinetic Energy > Value	Gas.ke [1 (p. 569)]
Equation Sources > Continuity > Variables > Velocity > Option	Cartesian Vector Components
Equation Sources > Continuity > Variables > Velocity > U	Liquid.Velocity u [1 (p. 569)]

Setting	Value
Equation Sources > Continuity > Variables > Velocity > V	Liquid.Velocity v [1 (p. 569)]
Equation Sources > Continuity > Variables > Velocity > W	Liquid.Velocity w [1 (p. 569)]

Footnote

1. Click the *Enter Expression* icon  to specify the CEL expression.

4. On the **Fluid Sources** tab, select `Liquid` in the list box, then select the **Liquid** check box and apply the following settings:

Setting	Value
Equation Sources	Continuity
Equation Sources > Continuity	(Selected)
Equation Sources > Continuity > Option	Fluid Mass Source
Equation Sources > Continuity > Source	-Liquid.WaFluxLG [1 (p. 570)]
Equation Sources > Continuity > MCF/Energy Sink Option	(Selected)
Equation Sources > Continuity > MCF/Energy Sink Option > Sink Option	Spec. Mass Frac. and Temp.
Equation Sources > Continuity > Variables > Temperature > Option	Value
Equation Sources > Continuity > Variables > Temperature > Value	SatT [1 (p. 570)]
Equation Sources > Continuity > Variables > Velocity > Option	Cartesian Vector Components
Equation Sources > Continuity > Variables > Velocity > U	0 [m s^-1]
Equation Sources > Continuity > Variables > Velocity > V	0 [m s^-1]
Equation Sources > Continuity > Variables > Velocity > W	0 [m s^-1]
Equation Sources	Energy
Equation Sources > Energy	(Selected)
Equation Sources > Energy > Option	Source
Equation Sources > Energy > Source	-Liquid.WaFluxLG*HtVapwa [1 (p. 570)]
Equation Sources > Energy > Source Coefficient	(Selected)
Equation Sources > Energy > Source Coefficient > Source Coefficient	-Liquid.vf*Liquid.density*Liquid.Cp/DtFalseMf [1 (p. 570)]

Footnote

1. Click the *Enter Expression* icon  to specify the CEL expression.

5. Click **OK**.

31.5.7. Creating Boundaries

This section outlines the steps to create the following boundaries: a `Gas Inlet` for the location where the steam is injected, an `Opening` boundary for the outer edges of the domain, and two symmetry boundaries. The wall of the injection pipe will assume the default boundary (a smooth, no-slip wall).

31.5.7.1. Inlet Boundary

At the gas inlet, create an inlet boundary that injects wet steam at a normal speed and static temperature set consistent with the problem description. The steam contains a liquid and vapor component whose sum of volume fractions is unity.

1. Create a new boundary named `Gas Inlet`.
2. Apply the following settings:

Tab	Setting	Value
Basic Settings	Boundary Type	Inlet
	Location	gas inlet
Boundary Details	Mass And Momentum > Option	Normal Speed
	Mass And Momentum > Normal Speed	341 [m s^-1]
	Turbulence > Option	Fluid Dependent
	Heat Transfer > Option	Static Temperature
	Heat Transfer > Static Temperature	373 K
Fluid Values	Boundary Conditions	Gas
	Boundary Conditions > Gas > Turbulence > Option	Low (Intensity = 1%)
	Boundary Conditions > Gas > Volume Fraction > Option	Value
	Boundary Conditions > Gas > Volume Fraction > Volume Fraction	1-0.45*0.4/1000 [1 (p. 571)]
	Boundary Conditions > Gas > Component Details	Steam3v
	Boundary Conditions > Gas > Component Details > Steam3v > Option	Mass Fraction

Tab	Setting	Value
	Boundary Conditions > Gas > Component Details > Steam3v > Mass Fraction	1
	Boundary Conditions	Liquid
	Boundary Conditions > Liquid > Volume Fraction > Option	Value
	Boundary Conditions > Liquid > Volume Fraction > Volume Fraction	0.45*0.4/1000 [1 (p. 571)]

Footnote

1. Click the *Enter Expression* icon  to specify the CEL expression.

3. Click **OK**.

31.5.7.2. Opening Boundary for the Outside Edges

For the outer edges of the domain, specify an opening boundary with a fixed pressure and flow direction. The direction specification is necessary to sufficiently constrain the velocity. At this opening boundary you need to set the temperature of air that may enter through the boundary. Set the opening temperature to be consistent with the problem description.

1. Create a new boundary named `Opening`.
2. Apply the following settings:

Tab	Setting	Value
Basic Settings	Boundary Type	Opening
	Location	air inlet,outer edge,outlet [1 (p. 572)]
Boundary Details	Mass And Momentum > Option	Opening Pres. and Dirn
	Mass And Momentum > Relative Pressure	0 [Pa]
	Flow Direction > Option	Normal to Boundary Condition
	Turbulence > Option	Medium (Intensity = 5%)
	Heat Transfer > Option	Opening Temperature
	Heat Transfer > Opening Temperature	25 [C] [2 (p. 572)]
Fluid Values	Boundary Conditions	Gas
	Boundary Conditions > Gas > Volume Fraction > Option	Value

Tab	Setting	Value
	Boundary Conditions > Gas > Volume Fraction > Volume Fraction	1
	Boundary Conditions > Gas > Component Details	Steam3v
	Boundary Conditions > Gas > Component Details > Steam3v > Option	Mass Fraction
	Boundary Conditions > Gas > Component Details > Steam3v > Mass Fraction	0.0
	Boundary Conditions	Liquid
	Boundary Conditions > Liquid > Volume Fraction > Option	Value
	Boundary Conditions > Liquid > Volume Fraction > Volume Fraction	0

Footnotes

1. Click the *Ellipsis* icon ... to open the selection dialog box, then select multiple items by holding the **Ctrl** key. Click **OK**.
2. Ensure that units are set to [C].

3. Click **OK**.

31.5.7.3. Creating the Symmetry Plane Boundaries

1. Create a new boundary named SymP1.
2. Apply the following settings:

Tab	Setting	Value
Basic Settings	Boundary Type	Symmetry
	Location	F29.26

3. Click **OK**.
4. Create a new boundary named SymP2.
5. Apply the following settings:

Tab	Setting	Value
Basic Settings	Boundary Type	Symmetry
	Location	F27.26

6. Click **OK**.

31.5.8. Creating a Time Step Expression

The conditions at each boundary determine the size of the time scale used in the time step. Generally, you can estimate an effective time step by dividing the displacement by the velocity at which a fluid is traveling. In this case, however, the velocity at the gas inlet approaches the speed of sound and the time step must be calculated by taking the height of the inlet and dividing it by the velocity at which the steam enters the system. The lower velocities at the outlet and opening boundaries allow the time step to be increased after the gas inlet properties have converged. Once all the values at the inlet, outlet, and openings have converged, a much larger time step is used to allow the overall solution to settle. In order to account for all these time step changes, an expression will be created.

Since the flow velocities are high at the jet inlet, you need to use a very small time step to capture the property variations at this location. The flow velocity decreases as you move away from the jet inlet, thus the time step can be increased systematically for better efficiency. You will now create a time step control expression called `Dtstep` that ramps up the time scale in stages:

1. Right-click `Expressions` in the **Outline** tree view and select **Insert** > **Expression**.
2. Set the name to `Dtstep` and click **OK**.
3. In the **Definition** area, type or copy and paste the following expression:

```
if (aitern <= 20, 1.0E-5[s], 5.0E-3 [s])
```

4. Click **Apply**.

31.5.9. Setting Solver Control

1. Click *Solver Control*  .
2. Apply the following settings:

Tab	Setting	Value
Basic Settings	Advection Scheme > Option	High Resolution
	Convergence Control > Max Iterations	1500
	Convergence Control > Fluid Timescale Control > Timescale Control	Physical Timescale
	Convergence Control > Fluid Timescale Control > Physical Timescale	Dtstep [1 (p. 574)]
	Convergence Criteria > Residual Type	RMS
	Convergence Criteria > Residual Target	1.0E-4
Advanced Options	Dynamic Model Control > Global Dynamic Model Control	(Selected)
	Multiphase Control	(Selected)

Tab	Setting	Value
	Multiphase Control > Volume Fraction Coupling	(Selected)
	Multiphase Control > Volume Fraction Coupling > Option	Segregated

Footnote

1. Click the *Enter Expression* icon  to specify the CEL expression.

3. Click **OK**.

31.5.10. Writing the CFX-Solver Input (.def) File

1. Click *Define Run*  .
2. In the **Write Solver Input File** dialog box, set **File name** to `SteamJet.def` and click **Save**.
3. If using Standalone Mode, quit CFX-Pre, saving the case (`.cfx`) file at your discretion.

31.6. Obtaining a Solution Using CFX-Solver Manager

When CFX-Pre has shut down and the CFX-Solver Manager has started, obtain a solution to the CFD problem by following the instructions below:

1. Ensure **Define Run** is displayed.

   CFX-Solver Input File should be set to `SteamJet.def`
2. Click **Start Run**.

   CFX-Solver runs and attempts to obtain a solution. This can take a long time depending on your system. Eventually a dialog box is displayed.
3. Note the number of iterations required to obtain a solution.
4. Select the check box next to **Post-Process Results** when the completion message appears at the end of the run.
5. If using Standalone Mode, select the check box next to **Shut down CFX-Solver Manager**.
6. Click **OK**.

31.7. Viewing the Results in CFD-Post

In this section, the steam molar fraction in the gas fluid, the gas to liquid and liquid to gas mass transfer rates, and the false time step will be illustrated on various regions.

31.7.1. Displaying the Steam Molar Fraction

1. Right-click a blank area in the viewer and select **Predefined Camera** > **View From -Z**.

   This ensures that the view is set to a position that is best suited to display the results.

2. From the menu bar, select **Insert** > **Contour**.
3. Under **Name**, type `Steam Molar Fraction` and click **OK**.
4. Apply the following settings:

Tab	Setting	Value
Geometry	Locations	SymP2
	Variable	Gas.Steam3v.Molar Fraction

5. Click **Apply**.

This will result in SymP2 shown colored by the molar fraction of steam. The steam enters at the `Gas Inlet` and therefore has a higher gas to liquid mass transfer rate at this location. You may want to zoom in near the gas inlet to view the molar fraction variation more closely.

31.7.2. Displaying the Gas to Liquid Mass Transfer Rate

1. In the tree view, clear the check box beside `Steam Molar Fraction` to hide it.
2. Right-click a blank area in the viewer and select **Predefined Camera** > **View From -Z**.

   This ensures that the view is set to a position that is best suited to display the results.
3. From the menu bar, select **Insert** > **Contour**.
4. Under **Name**, type `Gas to Liquid Flux` and click **OK**.
5. Apply the following settings:

Tab	Setting	Value
Geometry	Locations	SymP2
	Variable	Liquid.WaFluxGL

6. Click **Apply**.

This will result in SymP2 shown colored by the gas to liquid mass transfer rate. The steam enters at the `Gas Inlet` and therefore has a higher gas to liquid mass transfer rate at this location. You may want to zoom in near the gas inlet to view the mass transfer rate variation more closely.

31.7.3. Displaying the Liquid to Gas Mass Transfer Rate

1. In the tree view, clear the check box beside `Gas to Liquid Flux` to hide it.
2. Right-click a blank area in the viewer and select **Predefined Camera** > **View From -Z**.

   This ensures that the view is set to a position that is best suited to display the results.
3. From the menu bar, select **Insert** > **Contour**.
4. Under **Name**, type `Liquid to Gas Flux` and click **OK**.
5. Apply the following settings:

Tab	Setting	Value
Geometry	Locations	SymP2
	Variable	Liquid.WaFluxLG

6. Click **Apply**.

This will result in SymP2 shown colored by the liquid to gas mass transfer rate. The steam enters at the `Gas Inlet` and therefore has a higher liquid to gas mass transfer rate at this location. You may want to zoom in near the gas inlet to view the mass transfer rate variation more closely.

31.7.4. Displaying the Gas to Liquid and Liquid to Gas Phase Transfer Rates in Synchronous Views

1. In the viewer tool bar, open the viewport drop-down menu and click the option with two horizontal viewports.
2. In the viewer tool bar, click *Synchronize camera in displayed views*  .
3. Click a blank area in `View 2` so that it becomes the active view.
4. In the tree view, select the check box beside `Gas to Liquid Flux`.

Note

- You must disable *Synchronize camera in displayed views*  to allow separate contours to be displayed in each viewport
- Under `User Locations and Plots` in the tree view, ensure that only `Liquid to Gas Flux` is visible in `View 1`, and only `Gas to Liquid Flux` is visible in `View 2`.

5. In `View 2`, right-click a blank area in the viewer and select **Predefined Camera** > **View From -Z**.

   This ensures that the view is set to a position that is best suited to display the results.

You may want to zoom in near the gas inlet to view the differences between the gas to liquid and liquid to gas phase transfer rates.

31.7.5. Creating a Chart to Plot the False Time Step Along a Line

1. In the tree view, right-click `User Locations and Plots` and select **Insert** > **Location** > **Line**.
2. In the `Insert Line` dialogue box, use the default name and click **OK**.
3. Apply the following settings:

Tab	Setting	Value
Geometry	Definition > Method	Two Points
	Definition > Point 1	0, 0.005, 0.0002
	Definition > Point 2	5, 0.005, 0.0002
	Line Type > Cut	(Selected)

4. Click **Apply**.
5. From the menu bar, select **Insert** > **Chart**.
6. Name the chart `False Time Step` and click **OK**.
7. Apply the following settings:

Tab	Setting	Value
General	Title	False Time Step
Data Series	Data Source > Location	Line 1
X Axis	Data Selection > Variable	X
Y Axis	Data Selection > Variable	Liquid.FalseDt
	Axis Range > Logarithmic Scale	(Selected)

8. Click **Apply**.

The false time step peaks where the interphase mass transfer rate changes sign, and hence goes through zero. This is true because the false time step is inversely proportional to the absolute mass transfer rate.

When you have finished viewing the chart, quit CFD-Post.

Chapter 32: Modeling a Buoy using the CFX Rigid Body Solver

This tutorial includes:

32.1. Tutorial Features

In this tutorial you will learn about:

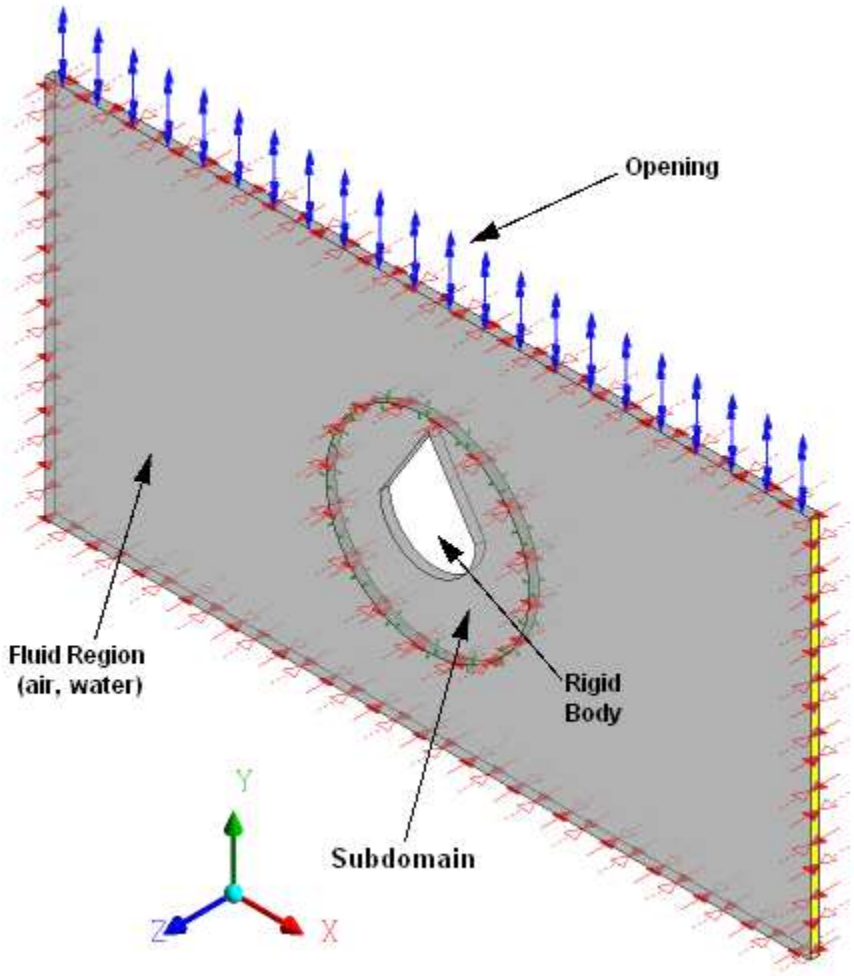
- Modeling a multiphase simulation in CFX-Pre.
- Creating and editing a rigid body in CFX-Pre.
- Creating and editing a subdomain in CFX-Pre.
- Creating a keyframe animation in CFD-Post.

Component	Feature	Details
CFX-Pre	User Mode	General Mode
	Analysis Type	Transient
	Fluid Type	General Fluid
	Domain Type	Single Domain
	Turbulence Model	Shear Stress Transport
	Heat Transfer	Isothermal
	Buoyant Flow	
	Multiphase	Homogeneous Model
	Rigid Body	3 degrees of freedom
	Boundary Conditions	Symmetry Plane
		Wall: No Slip
		Wall (Specified Displacement)
		Opening
		Mesh Motion option of Rigid Body Solution
	Subdomain	Mesh Motion option of Rigid Body Solution

Component	Feature	Details
	ANSYS CFX Command Language (CCL)	Importing Expressions
CFD-Post	Plots	Contour Plot
	Animations	Keyframe

32.2. Overview of the Problem to Solve

In this tutorial you will model the interaction between a rigid body (represented by a buoy) and two fluids (air and water) that make up the surrounding region, using a six degrees of freedom rigid-body solver. In this case, the fluid forces acting on the buoy cause motion that is constrained to three degrees of freedom: vertical and horizontal translation and rotation about an axis perpendicular to the translational directions. The motion of the floating buoy results from interactions between itself and the wave motion of the surrounding fluid created by an initial contraction of the domain in the X-direction.



The rigid body is surrounded by a fluid volume that is part air and part water (both at a static temperature of 25 °C). Because the rigid body has a density of 500 kg m^-3 – less than that of water (997 kg m^-3) – it floats atop the water's surface. The right-side wall, highlighted yellow in the image above, is given an initial velocity in the negative X-direction, thereby causing the fluid domain to shrink. This in turn causes waves in the water. An opening is required along the top face to allow air to move in and out of the fluid region

while the volume fraction of air and water are in a state of flux. Because of this contraction of the fluid region, you will also need to define the mesh motion of the domain, subdomain, and several of the boundary conditions.

A homogeneous, multiphase model will be used for this simulation because the air and water will maintain a well-defined interface. When setting up the initial conditions for the simulation, CCL-defined step functions will be used to determine the volume fractions of water and air defined by a function of height.

The relevant fluid parameters of this problem are:

- Density of water = `997 [kg m^-3]`
- Static temperature of water = `25 [C]`
- Density of air = `1.185 [kg m^-3]`
- Static temperature of air = `25 [C]`

The relevant physical parameters of the rigid body are:

- Mass = `39.39 [kg]`
- Density = `500 [kg m^-3]`
- Volume = `0.07878 [m^3]`
- Mass moment of inertia (XX, YY, ZZ, XY, XZ, YZ) = `(4.5, 2.1, 6.36, 0, 0, 0) [kg m^2]`
- Initial Center of Mass (X, Y, Z) = `(0, –0.1438, 0.05) [m]`

The first step in solving this problem is to import a pre-existing mesh file into CFX-Pre. A CCL file containing several mathematical expressions for this simulation will also be imported into CFX-Pre. The transient analysis conditions will then be defined and the default domain edited. A number of boundary conditions will also be created within CFX-Pre. Mesh motion within the domain and several of the boundary conditions will be specified because the domain will contract at the beginning of the simulation, and because the buoy will move freely due to wave motion causing motion of the fluids and hence the rigid body within the domain. In the first simulation, the motion of the mesh surrounding the rigid body will be fully coupled to the motion of the buoy, including the rotation of the buoy; the mesh will both rotate and translate with the buoy. In the second simulation the rotational and translational motion will be decoupled, allowing the inner cylindrical subdomain to rotate at the same rate as the buoy while the outer domain will deform solely with the translational motion of the buoy. In both simulations in CFD-Post, a contour plot will be created to visualize the air/water makeup of the fluid region and the mesh will be visible to observe the mesh when it deforms. In addition, one animation for each simulation will be produced in order to show the complex motion of the rigid body and mesh deformation.

32.3. Before You Begin

If this is the first tutorial you are working with, it is important to review the following topics before beginning:

- *Preparing a Working Directory and Starting ANSYS CFX in Standalone Mode* (p. 1)
- *Running ANSYS CFX Tutorials in ANSYS Workbench* (p. 2)
- *Changing the Display Colors* (p. 5)
- *Playing a Tutorial Session File* (p. 4)

32.4. Starting CFX-Pre

1. Copy the following files from the `<install_dir>\examples` directory into your working directory:

- `Buoy.pre`
- `Buoy_decoupled.pre`
- `Buoy.cfx`
- `Buoy.gtm`
- `Buoy.ccl`

2. Set the working directory in the CFX-Launcher and start CFX-Pre.

   For details, see *Preparing a Working Directory and Starting ANSYS CFX in Standalone Mode* (p. 1).

32.5. Rigid Body Simulation

This section describes the step-by-step definition of the flow physics in CFX-Pre.

If you want to set up the simulation automatically using a tutorial session file, run `Buoy.pre`. For details, see *Playing a Tutorial Session File* (p. 4). Then proceed to *Obtaining a Solution Using CFX-Solver Manager* (p. 596).

If you want to set up the simulation manually, proceed with the steps below.

1. In CFX-Pre, select **File** > **New Case**.
2. Select **General** and click **OK**.
3. Select **File** > **Save Case As**.
4. Under **File name**, type `Buoy`.
5. Click **Save**.

32.5.1. Importing the Mesh

1. Select **File** > **Import** > **Mesh**.
2. Apply the following settings:

Setting	Value
Files of type	CFX Mesh (*gtm *cfx)
File name	Buoy.gtm

3. Click **Open**.

32.5.2. Importing the Required Expressions From a CCL File

The mathematical expressions for this simulation will be imported from a CFX Command Language (CCL) file. These expressions will be used to set a monitor point and the physical parameters of the simulation: the fluid properties and the displacement of the walls and opening.

Note

The expressions or physics for a simulation can be saved to a CCL file at any time by selecting **File** > **Export** > **CCL**.

1. Select **File** > **Import** > **CCL**.

The **Import CCL** dialog box appears.

2. Under **Import Method**, select **Append**. This will start with the existing CCL already generated by CFX-Pre and append the imported CCL.

Note

Replace is useful if you have defined physics and want to update or replace them with newly-imported physics.

---

3. Select `Buoy.ccl`.
4. Click **Open**.
5. Double-click the `Expressions` section in the **Outline** tree to see a list of the expressions that have been imported.

   All expressions required for this simulation are displayed. Take a moment to look over each expression. A brief description of each expression will be provided wherever it is implemented within this tutorial.

6. Close the `Expressions` section by clicking *Close*  located at the top of the left workspace.

Note

Note that you could have entered these expressions manually into CFX-Pre by inserting new expressions and defining each with an appropriate formula.

32.5.3. Defining a Transient Simulation

1. In the **Outline** tree view, right-click **Analysis Type** and select **Edit**.
2. Apply the following settings:

Tab	Setting	Value
Basic Settings	Analysis Type > Option	Transient
	Analysis Type > Time Duration > Option	Total Time
	Analysis Type > Time Duration > Total Time	7.0 [s]
	Analysis Type > Time Steps > Option	Timesteps
	Analysis Type > Time Steps > Timesteps	0.025 [s] [1 (p. 583)]
	Analysis Type > Initial Time > Option	Automatic with Value
	Analysis Type > Initial Time > Time	0 [s]
1. A total time of `7.0 [s]` is implemented so that you get an adequate overview of the rigid body motion during post-processing. The `0.025 [s]`		

Tab	Setting	Value
	timestep provides enough detail in the solution without requiring an excessive amount of computation time for CFX-Solver.	

3. Click **OK**.

Note

You may ignore the physics validation messages regarding the lack of definition of transient results files and the lack of initial condition values. You will set up the transient results files and set the initial conditions later.

32.5.4. Editing the Domain

In this section you will create the fluid domain to reflect the multiphase, homogeneous region surrounding the buoy, define the fluids, and enable mesh motion.

1. Edit `Case Options` > `General` in the **Outline** tree view, ensure **Automatic Default Domain** and **Automatic Default Interfaces** are both selected, and click **OK**.
2. In the tree view, right-click `Default Domain`, select **Rename**, and set the new name to `buoy`.
3. In the tree view, right-click the newly renamed domain and select **Edit**.
4. Apply the following settings:

Tab	Setting	Value
Basic Settings	Location and Type > Location	`Assembly` [1 (p. 585)]
	Fluid and Particle Definitions	Delete `Fluid 1` [2 (p. 585)]
	Fluid and Particle Definitions	Create a new fluid named `Air at 25 C` [3 (p. 585)]
	Fluid and Particle Definitions	Create a new fluid named `Water at 25 C` [3 (p. 585)]
	Fluid and Particle Definitions > Air at 25 C > Material	Air at 25 C
	Fluid and Particle Definitions > Water at 25 C > Material	Water at 25 C [4 (p. 585)]
	Domain Models > Buoyancy Model > Option	Buoyant
	Domain Models > Buoyancy Model > Gravity X Dirn.	0 [m s^-2]
	Domain Models > Buoyancy Model > Gravity Y Dirn.	`-g` [5 (p. 586)]
	Domain Models > Buoyancy Model > Gravity Z Dirn.	0 [m s^-2]

Tab	Setting	Value
	Domain Models > Buoyancy Model > Buoy. Ref. Density	denAir [5 (p. 586)][6 (p. 586)]
	Domain Models > Mesh Deformation > Option	Regions of Motion Specified [7 (p. 586)]
	Domain Models > Mesh Deformation > Mesh Motion Model > Option	Displacement Diffusion [8 (p. 586)][9 (p. 586)]
	Domain Models > Mesh Deformation > Mesh Motion Model > Mesh Stiffness > Option	Value
	Domain Models > Mesh Deformation > Mesh Motion Model > Mesh Stiffness > Mesh Stiffness	1.0 [m^5 s^-1] / volcvol [5 (p. 586)][10 (p. 586)]
Fluid Models	Multiphase > Homogeneous Model	(Selected) [11 (p. 586)]
	Multiphase > Free Surface Model > Option	Standard
	Multiphase > Free Surface Model > Interface Compression Level	(Selected)
	Multiphase > Free Surface Model > Interface Compression Level > Interface Compression	2
	Heat Transfer > Homogeneous Model	(Selected)
	Turbulence > Option	Shear Stress Transport
Fluid Pair Models	Fluid Pair > Air at 25 C | Water at 25 C > Interphase Transfer > Option	Mixture Model [12 (p. 586)]
	Fluid Pair > Air at 25 C | Water at 25 C > Interphase Transfer > Interface Len. Scale	1.0 [mm]

1. Click *Multi-select from extended list* ... to open the **Selection Dialog** box, then select `Assembly` from this list. Click **OK**.
2. Ensure that you have `Fluid 1` selected and click *Remove Selected Item*  .
3. To create a new item, you must first click the *Add new item*  icon, then enter the name as required and click **OK**.
4. Click *Select from extended list* ... and then on *Import Library Data*  in the upper right corner of the resulting `Material` dialog box. When the `Select Library Data to Import` box appears, click *Expand*  located beside **Water Data**. Select `Water at 25 C` from the list. Click **OK**.

Tab	Setting	Value
5. In order to enter an expression, you must first click *Enter Expression*  .		
6. The buoyancy reference density is set to 1.185 kg/m^3, which is representative of air.		
7. This mesh deformation option enables you to specify the motion of the boundary mesh nodes using user-defined expressions created in the CFX Expression Language (CEL). These expressions of mesh motion are included in the CCL file that was imported at the beginning of the tutorial.		
8. To see the additional mesh motion settings, you may need to click *Roll Down*  located beside **Mesh Motion Model**.		
9. The Displacement Diffusion model for mesh motion preserves the relative mesh distribution of the initial mesh.		
10. The variable `volcvol` (volume of finite volumes) is a predefined variable related to the local mesh element volume. It is used here in the calculation of the mesh stiffness value. In this example, the mesh stiffness is set to be inversely proportional to `volcvol`, which results in higher stiffness in regions of smaller element size; these are the regions that are most probable to experience mesh folding.		
11. In a homogeneous, multiphase model, all fluids share a flow field, turbulence field, and so on. This is valid for models where the fluids have completely stratified; this is the case in this simulation.		
12. The mixture model is used for interphase transfer between the air and water. It is accurate for a pair of continuous fluids; this is the case in this simulation.		

5. Click **OK**.

32.5.5. Creating a Rigid Body

In this section you will specify the properties of a rigid body with three degrees of freedom: translation in the X- and Y-directions and rotation about the Z-axis. The rigid body definition will be applied to the wall boundary of the buoy to define the motion characteristics of the buoy. Further, you will specify the direction of gravity that acts upon the buoy's mass. Aside from gravity, no external forces are specified to act continuously on the buoy, however the motion of the buoy will be driven by fluid forces (from both the air and water) acting on the surface of the rigid body.

1. It is very important to give the rigid body a coordinate frame that is centered on its center of mass. Create a coordinate frame centered on the rigid body in its initial position, and oriented with its axes aligned with the rigid body axes/global coordinate axes as follows:

   a. Select **Insert** > **Coordinate Frame**.

   b. In the dialog box that appears, set **Name** to `RigidBodyCoordFrame` and click **OK**. The **Basic Settings** tab for the coordinate frame appears.

   c. Set **Option** to `Axis Points`.

   d. Set **Origin** to `0, -0.1438, 0.05`.

   e. Set **Z Axis Point** to `0, -0.1438, 1`.

f. Set **X-Z Plane Pt** to `1, –0.1438, 0.05`.

g. Click **OK**.

2. In the **Outline** tree view, right-click **Flow Analysis 1** and select **Insert** > **Rigid Body**.
3. Accept the default name, `Rigid Body 1`, by clicking **OK**.
4. Apply the following settings:

Tab	Setting	Value
Basic Settings	Mass	39.39 [kg] [1 (p. 588)]
	Location	BUOY
	Coord Frame	RigidBodyCoordFrame
	Mass Moment of Inertia > XX Component	4.5 [kg m^2]
	Mass Moment of Inertia > YY Component	2.1 [kg m^2]
	Mass Moment of Inertia > ZZ Component	6.36 [kg m^2]
	Mass Moment of Inertia > XY Component	0 [kg m^2]
	Mass Moment of Inertia > XZ Component	0 [kg m^2]
	Mass Moment of Inertia > YZ Component	0 [kg m^2]
Dynamics	Degrees of Freedom	(Selected)
	Degrees of Freedom > Translational Degrees of Freedom	(Selected)
	Degrees of Freedom > Translational Degrees of Freedom > Option	X and Y axes
	Degrees of Freedom > Rotational Degrees of Freedom	(Selected)
	Degrees of Freedom > Rotational Degrees of Freedom > Option	Z axis
	Gravity	(Selected)
	Gravity > Option	Cartesian Components
	Gravity > Gravity X Dirn.	0 [m s^-2]
	Gravity > Gravity Y Dirn.	-g [2 (p. 588)]
	Gravity > Gravity Z Dirn.	0 [m s^-2]
Initial Conditions	Center of Mass	(Selected)
	Center of Mass > Option	Automatic[3 (p. 588)]
	Linear Velocity	(Selected)
	Linear Velocity > Option	Automatic with Value

Tab	Setting	Value
	Linear Velocity > X Component	0 [m s^-1]
	Linear Velocity > Y Component	0 [m s^-1]
	Linear Velocity > Z Component	0 [m s^-1]
	Angular Velocity	(Selected)
	Angular Velocity > Option	Automatic with Value
	Angular Velocity > X Component	0 [radians s^-1]
	Angular Velocity > Y Component	0 [radians s^-1]
	Angular Velocity > Z Component	0 [radians s^-1]

1. The values in this table are taken directly from the problem description found in the *Overview of the Problem to Solve* (p. 580) section.
2. In order to enter an expression, you must first click *Enter Expression*  .
3. Setting this option to **Automatic** defaults the center of mass of the rigid body to the origin of the `RigidBodyCoordFrame`. In most cases, this will be the correct setting.

5. Click **OK**.

32.5.6. Creating the Boundary Conditions

In this section symmetry boundaries will be created for the front and back planes of the given geometry; this is required because a 2D representation of the flow field is being modeled. Wall boundaries will also be created for the bottom, stationary side, moving side, and buoy body sections of the fluid region. Because the right-side wall will be provided with an initial velocity in the negative X-direction, you will define mesh motion along this direction for the moving wall boundaries. An opening boundary will also be created along the top of the fluid region to allow air to flow freely in and out of this region because of the interactions between the air and water (as a result of the moving wall).

32.5.6.1. Symmetry Boundaries

The front and back planes each require a symmetry boundary.

1. Create a new boundary named `back`.
2. Apply the following settings:

Tab	Setting	Value
Basic Settings	Boundary Type	Symmetry
	Location	BACK A, BACK B [1 (p. 588)]
Boundary Details	Mesh Motion > Option	Unspecified [2 (p. 589)]

1. Hold the **Ctrl** key while selecting both BACK A and BACK B from the list.

Tab	Setting	Value
2. In the unspecified mesh motion option, no mesh motion constraints are applied directly to the nodes. Instead, mesh motion is governed by the constraints in other regions of the mesh.		

3. Click **OK**.
4. Create a second boundary named `front`.
5. Apply the following settings:

Tab	Setting	Value
Basic Settings	Boundary Type	Symmetry
	Location	FRONT A, FRONT B [1 (p. 589)]
Boundary Details	Mesh Motion > Option	Unspecified[2 (p. 589)]
1. Hold the **Ctrl** key while selecting both FRONT A and FRONT B from the list. 2. In the unspecified mesh motion option, no mesh motion constraints are applied directly to the nodes. Instead, mesh motion is governed by the constraints in other regions of the mesh.		

6. Click **OK**.

32.5.6.2. Wall Boundaries

The top, bottom, and sides of the fluid region all require wall boundaries. In addition, the surface between the fluid region and the rigid body, `Rigid Body 1`, requires a wall boundary; this wall boundary will move according to the rigid body solution.

1. Create a new boundary named `Buoy Surface`.
2. Apply the following settings:

Tab	Setting	Value
Basic Settings	Boundary Type	Wall
	Location	BUOY
Boundary Details	Mesh Motion > Option	Rigid Body Solution
	Mesh Motion > Rigid Body	Rigid Body 1
	Mass and Momentum > Option	No Slip Wall
	Wall Roughness > Option	Smooth Wall

3. Click **OK**.
4. Create a new boundary named `wall`.
5. Apply the following settings:

Tab	Setting	Value
Basic Settings	Boundary Type	Wall
	Location	BOTTOM, S1, S2 [1 (p. 590)]
Boundary Details	Mesh Motion > Option	Specified Displacement
	Mesh Motion > Displacement > Option	Cartesian Components
	Mesh Motion > Displacement > X Component	wallMeshMot [2 (p. 590)][3 (p. 590)]
	Mesh Motion > Displacement > Y Component	0.0 [m] [4 (p. 590)]
	Mesh Motion > Displacement > Z Component	0.0 [m]
	Mass and Momentum > Option	No Slip Wall

1. Hold the **Ctrl** key while selecting `BOTTOM`, `S1`, and `S2` from the list.
2. In order to enter an expression, you must first click *Enter Expression*  .
3. This displacement expression, defined in the CCL, will allow the mesh nodes to shift in the X-direction because the fluid domain is compressed in this direction during the simulation. The X-coordinate runs from -4.0 m to 4.0 m, but this mesh motion along the bottom of the fluid region only occurs when the X-coordinate is between 0 m and 4.0 m. Thus, `wallMeshMot` utilizes a simple logical expression to apply the mesh motion only when the X-coordinate is greater than or equal to 0 m.

   The second displacement expression that is present in the CCL, `wxdisp`, allows the mesh nodes to shift in the X-direction, because the right-side wall will be moving and compressing the fluid region during the simulation. It is defined in the CCL as a logical expression that remains steady at `0.0 [m]` until it is turned "on" at time `tOn`, at which point a time-dependent, sinusoidally increasing displacement is applied. This displacement increases until 1.0 seconds into the simulation, when it plateaus at this final value. The expression `wallMeshMot` is dependent on `wxdisp`, as can be seen in the CCL, and will therefore take the motion of the moving side wall into account. This expression will have no effect on the stationary wall because this wall has an X-coordinate that is outside the part of the expression that specifies a displacement. Thus, `wallMeshMot` can be applied to all three of these walls; it is equally applicable to each.
4. The left-side of the fluid region maintains its position throughout the simulation and it is necessary to define mesh deformation of the bottom in the X-direction only. Therefore, set the mesh displacement in the Y- and Z-directions to `0.0 [m]`.

6. Click **OK**.

32.5.6.3. Opening Boundary

1. Create a new boundary named `top`.

2. Apply the following settings:

Tab	Setting	Value
Basic Settings	Boundary Type	Opening
	Location	TOP
Boundary Details	Mesh Motion > Option	Specified Displacement
	Mesh Motion > Displacement > Option	Cartesian Components
	Mesh Motion > Displacement > X Component	wallMeshMot [1 (p. 591)]
	Mesh Motion > Displacement > Y Component	0.0 [m]
	Mesh Motion > Displacement > Z Component	0.0 [m]
	Mass and Momentum > Option	Opening Pres. and Dirn
	Mass and Momentum > Relative Pressure	0 [Pa]
Fluid Values	Boundary Conditions > Air at 25 C > Volume Fraction > Volume Fraction	1.0
	Boundary Conditions > Water at 25 C > Volume Fraction > Volume Fraction	0.0 [2 (p. 591)]

1. The same mesh motion is provided for the `top` boundary and the `bottom` boundary. They will move in unison.
2. The `top` boundary comes into contact only with air, and not with water. The volume fraction of the opening for air is set to 1.0 and that of water to 0.0, thus allowing only air to pass through the opening.

3. Click **OK**.

Note

Opening boundary types are used to allow the flow to leave and re-enter the domain. This behavior is expected due to the motion of the water and the interaction between the air and water in the fluid region.

32.5.7. Setting Initial Values

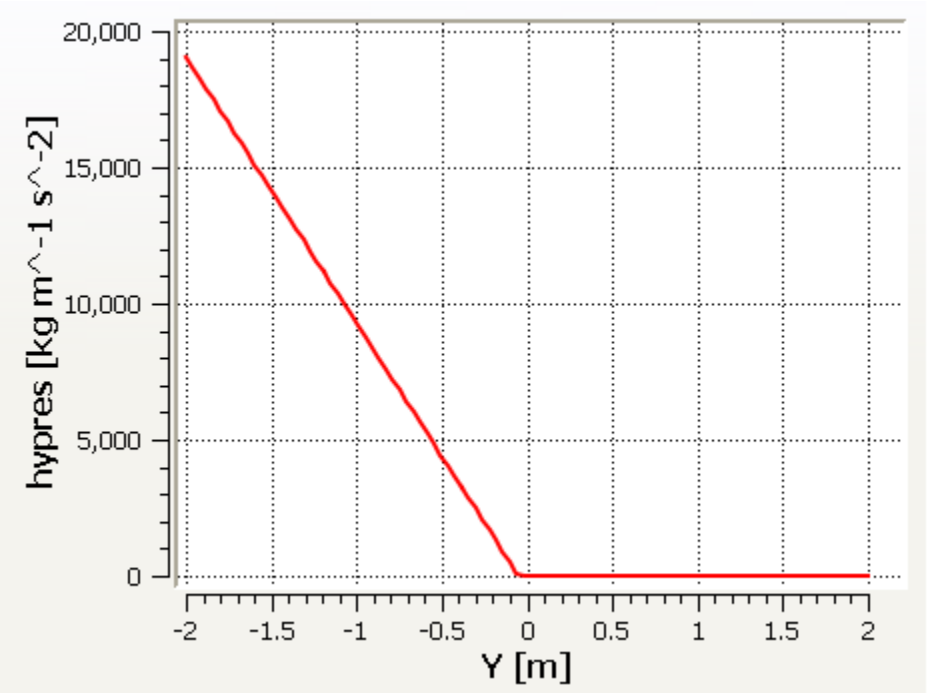
Because a transient simulation is being modeled, initial values are required for all variables.

1. Click *Global Initialization*  .
2. Apply the following settings:

Tab	Setting	Value
Global Settings	Initial Conditions > Cartesian Velocity Components > U	0 [m s^-1]
	Initial Conditions > Cartesian Velocity Components > V	0 [m s^-1]
	Initial Conditions > Cartesian Velocity Components > W	0 [m s^-1]
	Initial Conditions > Static Pressure > Option	Automatic with Value
	Initial Conditions > Static Pressure > Relative Pressure	hypres [1 (p. 592)][2 (p. 592)]
	Initial Conditions > Turbulence > Option	Intensity and Eddy Viscosity Ratio
Fluid Settings	Fluid Specific Initialization > Air at 25 C > Initial Conditions > Volume Fraction > Volume Fraction	airvol [1 (p. 592)]
	Fluid Specific Initialization > Water at 25 C > Initial Conditions > Volume Fraction > Volume Fraction	watvol [1 (p. 592)][3 (p. 593)]

1. In order to enter an expression, you must first click *Enter Expression*  .
2. The expression `hypres` is defined in the CCL file and gives the relative pressure as a function of the volume of water in the fluid region and the vertical distance (Y-direction value). The expression results in an initial pressure gradient that linearly decreases when the value of the Y-coordinate increases (or the water depth decreases). When the Y-coordinate reaches the interface between the air and water, the initial pressure plateaus at a value of zero (due to the reliance of the expression `hypres` on `watvol`, which is explained in greater detail in the next footnote). The initial pressure gradient produced by this expression can be observed in the image below.

Tab	Setting	Value



To create this plot in CFX-Pre:

a. Double-click **Expressions** in the **Outline** tree view.

b. Double-click the `hypres` expression.

c. Select the **Plot** tab under `Details of` **hypres**.

d. Select `Expression Variables: Y`.

e. Set the `Start of Range` to `-2.0 [m]` and `End of Range` to `2.0 [m]`.

f. Click **Plot Expression**.

3. The `airvol` and `watvol` expressions were provided in the CCL file. The expression `watvol` is a step function that returns a value of `1.0` for all Y-coordinate values less than the initial water height (0.05 [m]), `0.0` for all Y-coordinate values greater than the initial water height, and `0.5` when the Y-coordinate value is equal to the initial water height.

3. Click **OK**.

32.5.8. Setting the Solver Control

In this section, you will adjust the solver control settings to promote a quicker solution time and to enable the frequency of when the rigid body solver is executed.

1. Click *Solver Control*  .

2. Apply the following settings:

Tab	Setting	Value
Equation Class Settings	Equation Class > Mesh Displacement	(Selected)
	Equation Class > Mesh Displacement > Convergence Control	(Selected)
	Equation Class > Mesh Displacement > Convergence Control > Max. Coeff. Loops	4 [1 (p. 594)]
	Equation Class > Mesh Displacement > Convergence Control > Min. Coeff. Loops	2
Rigid Body Control	Rigid Body Control	(Selected)
	Rigid Body Control > Rigid Body Solver Coupling Control > Update Frequency	Every Coefficient Loop [2 (p. 594)]
	Rigid Body Control > Angular Momentum Equation Control	(Selected) [3 (p. 594)]
Advanced Options	Multiphase Control	(Selected)
	Multiphase Control > Initial Volume Fraction Smoothing	(Selected)
	Multiphase Control > Initial Volume Fraction Smoothing > Option	Volume-Weighted [4 (p. 594)]

1. The maximum number of coefficient loops is set to 4 and the minimum number of coefficient loops to 2 to ensure that the solver completes at least 2 loops per timestep, and no more than 4. In this simulation it will ensure a relatively resolved and accurate solution within a short period of time.
2. By setting the update frequency to every coefficient loop you are specifying that CFX-Solver will call the rigid body solver during every coefficient loop within each timestep. This may increase total solution time, however the motion of the rigid body will be better resolved.
3. This sets the integration scheme for the angular momentum equations to the second-order Simo Wong scheme, which is robust and energy-conserving.
4. If the initial conditions for volume fraction have a discontinuity, startup robustness problems may occur. Choosing volume-weighted smoothing of these volume fractions may improve startup robustness.

3. Click **OK**.

32.5.9. Setting the Output Control

In this section, you will set transient results for selected variables to be captured every three timesteps. You will also create two monitor points so that you can track the progress in CFX-Solver Manager.

1. Click *Output Control* .
2. Click the **Trn Results** tab.

3. In the **Transient Results** editor, click *Add new item*  , set **Name** to `Transient Results 1`, and click **OK**.
4. Apply the following settings to `Transient Results 1`:

Setting	Value
Transient Results 1 > Option	Selected Variables
Transient Results 1 > Output Variables List	Pressure, Total Mesh Displacement, Velocity, Water at 25 C.Volume Fraction [1 (p. 595)]
Transient Results 1 > Output Frequency > Option	Time Interval
Transient Results 1 > Output Frequency > Time Interval	tOn[2 (p. 595)]
1. Click *Multi-select from extended list* ... beside the entry box, and make multiple selections in the **Output Variables List** by holding down the **Ctrl** key and clicking on the required variables. 2. In order to enter an expression, you must first click *Enter Expression*  .	

5. Click the **Monitor** tab and apply the following settings:

Monitor Objects	(Selected)
Monitor Objects > Monitor Points and Expressions	Create a new Monitor Point and enter the name `Buoy Force` [1 (p. 595)][2 (p. 595)].
Monitor Objects > Monitor Points and Expressions > Buoy Force > Option	Expression
Monitor Objects > Monitor Points and Expressions > Buoy Force > Expression Value	force_y()@Buoy Surface
Monitor Objects > Monitor Points and Expressions	Create a new Monitor Point and enter the name `Buoy Torq` [1 (p. 595)][3 (p. 595)].
Monitor Objects > Monitor Points and Expressions > Buoy Torq > Option	Expression
Monitor Objects > Monitor Points and Expressions > Buoy Torq > Expression Value	torque_z()@Buoy Surface
1. To create a new item, you must first click the *Add new item* icon, then enter the name as required and click **OK**. 2. This monitor point will track the force acting on the `rigid body` in the Y-direction. 3. This monitor point will track the torque of the `rigid body` relative to the Z-axis.	

6. Click **OK**.

32.5.10. Writing the CFX-Solver Input (.def) File

1. Click *Define Run* .
2. Apply the following settings:

Setting	Value
File name	Buoy.def

3. Click **Save**.
4. CFX-Solver Manager automatically starts and, on the **Define Run** dialog box, the **Solver Input File** is set.
5. Quit CFX-Pre, saving the simulation (`.cfx`) file.

32.5.11. Obtaining a Solution Using CFX-Solver Manager

When CFX-Pre has shut down and the CFX-Solver Manager has started, obtain a solution to the CFD problem by following the instructions below.

1. Ensure **Define Run** is displayed.

   Solver Input File should be set to `Buoy.def`.
2. Click **Start Run**.

   CFX-Solver runs and attempts to obtain a solution. This can take a long time, depending on your system. Eventually a dialog box is displayed.

Note

After the CFX-Solver Manager has run for a short time, you can track the monitor points you created in CFX-Pre by clicking the `User Points` tab that appears at the top of the graphical interface of CFX-Solver Manager. The two monitor points — Buoy Force and Buoy Torq — are monitored in the global coordinate frame and not the coordinate frame attached to the buoy. You can also view the level of convergence of the rigid body solution through the `Rigid Body Convergence` tab. Finally, the rigid body position and Euler angles can be displayed by going to the main menu and selecting **Monitors** > **Rigid Body** > **Rigid Body Position** and **Monitors** > **Rigid Body** > **Rigid Body Euler Angles**, respectively.

Note

New monitor points can be toggled within the current plot by right clicking on the plot and selecting **Monitor Properties**. A window will display available plot line variables. Select the box to the left of the property to display it — the plot will adjust the scale so that all properties appear.

3. Select the check box next to **Post-Process Results** when the completion message appears at the end of the run.
4. If using Standalone Mode, select the check box next to **Shut down CFX-Solver Manager**.

5. Click **OK**.

32.5.12. Viewing the Results in CFD-Post

In this section, you will create a contour plot for this case. An animation will then be created to show the movement of the rigid body in the fluid domain. Furthermore, the minimum face angle of the mesh will be calculated for comparison purposes between the simulations.

32.5.12.1. Creating a Contour Plot

1. Right-click a blank area in the viewer and select **Predefined Camera** > **View From +Z**.
2. Create a new **Plane** and accept the default name.
3. Apply the following settings:

Tab	Setting	Value
Geometry	Definition > Method	XY Plane
	Definition > Z	0.05 [m]

4. Click **Apply**.
5. Create new contour and accept the default name.
6. Apply the following settings:

Tab	Setting	Value
Geometry	Locations	Plane 1
	Variable	Water at 25 C.Volume Fraction
	Color Map	White to Blue
	# of Contours	10

7. Click **Apply**.
8. Select **File** > **Save State** and choose the name `Buoy.cst`.
9. Click **Save**.

32.5.12.2. Creating a Keyframe Animation

A short animation of the rigid body and surrounding fluid region, starting from rest and given an initial velocity, will be created to show the complex motion of the rigid body and deformation of the mesh because of the waves created in the fluid region. You will record a short animation that can be played in an MPEG player.

1. Ensure that `Contour 1` is visible in the 3D Viewer (make sure there is a check mark beside `Contour 1` in the **Outline** tree view).
2. Turn off the visibility of `Plane 1` and `Default Legend View 1` to better see the movement of the buoy.
3. Edit `Wireframe` and apply the following settings:

Tab	Setting	Value
Definition	Show surface mesh	(Selected)

4. Click **Apply**.
5. Click the *Timestep Selector* in the toolbar. Select the 1st time step and click **Apply**.
6. Click *Animation* in the *Timestep Selector* dialog box.
7. In the **Animation** dialog box, select the **Keyframe Animation** option.
8. Click *New* to create `KeyframeNo1`.
9. Select `KeyframeNo1`, then set **# of Frames** to 93, then press **Enter** while the cursor is in the **# of Frames** box.

Tip

Be sure to press **Enter** and confirm that the new number appears in the list before continuing.

10. In the **Timestep Selector**, select time step 280 and click **Apply**.
11. In the **Animation** dialog box, click *New* to create `KeyframeNo2`.
12. Ensure that *More Animation Options* is pushed down to show more animation settings.
13. Select **Loop**.
14. Ensure that *Repeat forever* (next to **Repeat**) is not selected (not pushed down).
15. Select **Save Movie**.
16. Set **Format** to `MPEG1`.
17. Prepare to save the movie file:
    a. If you want to save the file in your working directory, set **File name** to `Buoy.mpg` in the text box beside **Save Movie**.
    b. If you want to save the file somewhere other than your working directory, click *Browse* (next to **Save Movie**) set the new path and movie file name. Click **Save**.

    The movie file name (including the path) has been set, but the animation has not yet been produced.
18. Click *To Beginning*.

    This ensures that the animation will begin at the first keyframe.
19. After the first keyframe has been loaded, click *Play the animation*.

- The MPEG will be created as the animation proceeds.
- This will be slow, since results will be loaded and objects will be created for each time step.
- To view the movie file, you need to use a viewer that supports the MPEG format.

Note

To explore additional animation options, click the **Options** button. On the **Advanced** tab of the **Animation Options** dialog box, there is a **Save Frames As Image Files** check box. By selecting this check box, the JPEG or PPM files used to encode each frame of the movie will persist after movie creation; otherwise, they will be deleted.

20. Close the **Animation** dialog box when the animation is complete.

32.5.12.3. Calculating the Minimum Mesh Face Angle

In this step, you will calculate the Minimum Face Angle of the mesh which is an indicator of the overall mesh quality during the deformation of the mesh. A Minimum Face Angle of greater than 15° is one indicator of a good quality mesh. However an angle of between 10° and 15° is also acceptable but may produce inaccuracies in that region of the mesh during the simulation.

1. Click the *Timestep Selector*  , select the 162nd time step and click **Apply**.
2. Select **Tools** > **Mesh Calculator** or click the **Calculators** tab and select **Mesh Calculator**.
3. Apply the following settings

Tab	Setting	Value
Mesh Calculator	Function	Minimum Face Angle

4. Click **Calculate**.
5. When you have finished, close the **Timestep Selector** dialog box and exit from CFD-Post.

The 162nd timestep was chosen arbitrarily to contrast the mesh quality between this simulation and the following one. In this simulation the Minimum Face Angle during the 162nd time step is approximately 13°.

32.6. Rigid Body Simulation with Decoupled Mesh Motion

In this section you will use a subdomain to decouple rotation (that is, to allow independent rotation of the mesh in each part of the domain). Furthermore, you will edit the domain interface boundaries to restrict mesh deformation on the outer part of the domain to translational only; the inner cylindrical part of the domain will rotate and translate at the same rate as the buoy.

If you want to set up the simulation automatically using a tutorial session file, run `Buoy_decoupled.pre`. For details, see *Playing a Tutorial Session File* (p. 4). Then proceed to *Obtaining a Solution for the Simulation with Decoupled Motion Using CFX-Solver Manager* (p. 601).

If you want to set up the simulation manually, proceed to the following section.

32.6.1. Opening an Existing Simulation

1. Start CFX-Pre if it is not already running.
2. Select **File** > **Open Case**.
3. From your working directory, select `Buoy.cfx` and click **Open**.
4. Select **File** > **Save Case As**.

5. Set **File name** to `Buoy_decoupled.cfx`.
6. Click **Save**.

32.6.2. Creating a Subdomain

The subdomain, domain interfaces, and buoy must share common rigid body characteristics. This is necessary because the inner cylinder and the rigid body must translate and rotate at the same rate to properly isolate the motions. All relative movement between the inner cylinder and the buoy will be eliminated, causing zero mesh deformation within the inner cylinder. A subdomain is not strictly necessary to decouple rotational motions. However, the subdomain increases the robustness of the simulation by ensuring that the entire mesh within the inner cylinder has the same physical properties as the rigid body, not just at the buoy boundary and inner cylinder domain interface (this will be set up in the next step).

1. Select **Insert** > **Subdomain** from the main menu or click *Subdomain*  .
2. Set the subdomain name to `rot_trans` and click **OK**.
3. Apply the following settings:

Tab	Setting	Value
Basic Settings	Location	B86
Mesh Motion	Mesh Motion > Option	Rigid Body Solution
	Mesh Motion > Rigid Body	Rigid Body 1
	Mesh Motion > Motion Constraints	(Selected)
	Mesh Motion > Motion Constraints > Motion Constraints	None

4. Click **OK**.

32.6.3. Editing the Domain Interfaces

You will edit the domain interfaces to restrict the rotational movement of the mesh surrounding the subdomain. The mesh that is located on the inner cylindrical domain interface will be assigned the same physical properties as that of the rigid body.

1. Edit `Simulation` > `Flow Analysis 1` > `buoy` > `Default Fluid Fluid Interface Side 1`.
2. Apply the following settings:

Tab	Setting	Value
Basic Settings	Location	F74.27[1]
Boundary Details	Mesh Motion > Option	Rigid Body Solution
	Mesh Motion > Rigid Body	Rigid Body 1
	Mesh Motion > Motion Constraints	(Selected)
	Mesh Motion > Motion Constraints > Motion Constraints	Ignore Rotations[2]

1. This is the outer cylindrical domain interface.

Tab	Setting	Value
2. Ignore Rotations constrains the outer domain interface to only translational motion.		

3. Click **OK**.
4. Edit `Simulation` > `Flow Analysis 1` > `buoy` > `Default Fluid Fluid Interface Side 2`.
5. Apply the following settings:

Tab	Setting	Value
Basic Settings	Location	F89.86[1]
Boundary Details	Mesh Motion > Option	Rigid Body Solution
	Mesh Motion > Rigid Body	Rigid Body 1
	Mesh Motion > Motion Constraints	(Selected)
	Mesh Motion > Motion Constraints > Motion Constraints	None
1. This is the inner cylindrical domain interface.		

6. Click **OK**.

32.6.4. Writing the CFX-Solver Input (.def) File

1. Click *Define Run*  .
2. Apply the following settings:

Setting	Value
File name	Buoy_decoupled.def

3. Click **Save**.
4. CFX-Solver Manager automatically starts and, on the **Define Run** dialog box, the **Solver Input File** is set.
5. Quit CFX-Pre, saving the simulation (`.cfx`) file.

32.6.5. Obtaining a Solution for the Simulation with Decoupled Motion Using CFX-Solver Manager

When CFX-Pre has shut down and the CFX-Solver Manager has started, obtain a solution to the CFD problem by following the instructions below.

1. Ensure **Define Run** is displayed.

   Solver Input File should be set to `Buoy_decoupled.def`.
2. Click **Start Run**.

CFX-Solver runs and attempts to obtain a solution. This can take a long time depending on your system. Eventually a dialog box is displayed.

3. Select the check box next to **Post-Process Results** when the completion message appears at the end of the run.
4. If using Standalone Mode, select the check box next to **Shut down CFX-Solver Manager**.
5. Click **OK**.

32.6.6. Viewing the Results in CFD-Post

In this section, you will create a contour plot of `Water at 25 C.Volume Fraction` to show the water content of the fluid region, and to illustrate the interface between the air and water. An animation will then be created to show the movement of the rigid body in the fluid domain. Furthermore, the minimum face angle of the mesh will be calculated for comparison purposes between the simulations.

32.6.6.1. Loading a Contour Plot from the State File

In the first part of the tutorial, you created a plane and a contour plot, then saved a state file named `Buoy.cst`. You will load the resulting state file so that you do not have to create a new plane and contour plot:

1. Select **File** > **Load State** and choose the name `Buoy.cst`.
2. Click **Open**.

32.6.6.2. Creating a Keyframe Animation

A short animation of the rigid body and surrounding fluid region, starting from rest and given an initial velocity, will be created to show the complex motion of the rigid body and deformation of the mesh because of the waves created in the fluid region. You will record a short animation that can be played in a MPEG player.

1. Ensure that `Contour 1` is visible in the 3D Viewer (make sure there is a check mark beside `Contour 1` in the **Outline** tree view).
2. Turn off the visibility of `Plane 1` and `Default Legend View 1` to better see the movement of the buoy.
3. Edit `Wireframe` and apply the following settings:

Tab	Setting	Value
Definition	Show surface mesh	(Selected)

4. Click the *Timestep Selector*  . Select the 1st time step and click **Apply**.
5. Click *Animation*  in the *Timestep Selector* dialog box.
6. In the **Animation** dialog box, select the **Keyframe Animation** option.
7. Click *New*  to create `KeyframeNo1`.
8. Select `KeyframeNo1`, then set **# of Frames** to 93, then press **Enter** while the cursor is in the **# of Frames** box.

Tip

Be sure to press **Enter** and confirm that the new number appears in the list before continuing.

9. Use the **Timestep Selector** to load the 280th time step.
10. In the **Animation** dialog box, click *New*  to create `KeyframeNo2`.
11. Ensure that *More Animation Options*  is pushed down to show more animation settings.
12. Select **Loop**.
13. Ensure that *Repeat forever*  (next to **Repeat**) is not selected (not pushed down).
14. Select **Save Movie**.
15. Set **Format** to `MPEG1`.
16. Set **File name** to `Buoy_decoupled.mpg`.
17. If you want to save the animation to a location other than your working directory, click *Browse*  (next to **Save Movie**) to set the path to a different directory and click **Save**.

    The movie file name (including the path) has been set, but the animation has not yet been produced.
18. Click *To Beginning*  .

    This ensures that the animation will begin at the first keyframe.
19. After the first keyframe has been loaded, click *Play the animation*  .

    - The MPEG will be created as the animation proceeds.
20. Close the **Animation** dialog box when the animation is complete.

32.6.6.3. Calculating the Minimum Mesh Face Angle

In this section you will calculate the Minimum Face Angle of the mesh which is an indicator of the overall mesh quality during the deformation of the mesh. A Minimum Face Angle of greater than 15° is one indicator of a good quality mesh. However an angle of between 10° and 15° is also acceptable but may produce inaccuracies in that region of the mesh during the simulation.

1. Click the *Timestep Selector*  and load the 162nd time step.
2. Select **Tools** > **Mesh Calculator** or click the **Calculators** tab and select **Mesh Calculator**.
3. Apply the following settings

Tab	Setting	Value
Mesh Calculator	Function	Minimum Face Angle

4. Click **Calculate**.

   You can also check other time steps to calculate the mesh quality throughout the simulation.

5. When you have finished, close the **Timestep Selector** dialog box.

32.7. Comparing the Two Cases

In this section you will compare two cases. First, compare the animations:

1. With `Buoy_decoupled_001.res` already loaded in CFD-Post, select **File** > **Load Results**.
2. In the **Load Results File** dialog box, select **Keep current cases loaded**, then select the file `Buoy_001.res`. Click **Open**.
3. Click the viewport icon  and select  .
4. Click the synchronize active view icon  .
5. Right-click within the 3D view and select **Predefined Camera** > **View from +Z** to orient the view. Click *Fit View*  to scale the buoy appropriately within the 3D viewer.
6. In the Outline tree, double-click **Case Comparison**.
7. In the **Case Comparison** editor, select **Case Comparison Active**, then ensure that both cases are set to `Current Step: 0`. Click **Apply**.
8. Ensure that in each of the views:
   - `Contour 1` is visible
   - The plane and default legend are hidden
   - `Wireframe` has **Show surface mesh** selected

   Note

   To show/hide plots, toggle the check box next to the plot name in the **Outline** tree view. This toggles the visibility of the plot in the currently selected view. Because the synchronization of active views has been enabled, this also modifies the visibility of all other views to match the currently selected view.

9. Click the *Timestep Selector*  .
10. Click *Animation*  in the *Timestep Selector* dialog box.
11. In the **Animation** dialog box, select the **Keyframe Animation** option.
12. Delete the two existing Keyframes using the delete icon  , because they display the results from only one results file. Then set up the Keyframe Animation in the same way as for the animations you created previously in this tutorial.
13. Beside **Save Movie**, set the movie file name to `Buoy_comparison.mpg`.
14. Click *Stop*  , then click *To Beginning*  .

    This ensures that the animation will begin at the first keyframe.
15. After the first keyframe has been loaded, click *Play the animation*  .
    - The MPEG will be created as the animation proceeds.

16. Close the **Animation** dialog box when the animation is complete.

Now compare the mesh deformations:

1. Click the *Timestep Selector* and load the 162nd time step.

   The top two views show the differences in the mesh deformation.
2. Select **Tools** > **Mesh Calculator** or click the **Calculators** tab and select **Mesh Calculator**.
3. Apply the following settings

Tab	Setting	Value
Mesh Calculator	Function	Minimum Face Angle

4. Click **Calculate**. Results from both cases appear.

   You can also check other time steps to calculate the mesh quality throughout the simulation.
5. When you have finished, close the **Timestep Selector** dialog box and exit from CFD-Post.

Index

Symbols

A

B

C

D

E

F

P

R

S

T

U

V

W