

ANSYS CFX Tutorials

ANSYS CFX Release 11.0

December 2006

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

Copyright and Trademark Information

© 1996-2006 ANSYS Europe, Ltd. All rights reserved. Unauthorized use, distribution, or duplication is prohibited.

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

Disclaimer Notice

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

ANSYS, Inc. and ANSYS Europe, Ltd. are UL registered ISO 9001:2000 companies.

U.S. Government Rights

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

Third-Party Software

See the online documentation in the product help files for the complete Legal Notice for ANSYS proprietary software and third-party software. The ANSYS third-party software information is also available via download from the Customer Portal on the ANSYS web page. If you are unable to access the third-party legal notices, please contact ANSYS, Inc.

Published in the U.S.A.

Table of Contents

Copyright and Trademark Information

Disclaimer Notice

U.S. Government Rights

Third-Party Software

Introduction to the ANSYS CFX Tutorials

Overview	1
Setting the Working Directory	1
Changing the Display Colors	2

Tutorial 1: Simulating Flow in a Static Mixer Using CFX in Standalone Mode

Introduction	3
Before You Begin	4
Tutorial 1 Features	4
Overview of the Problem to Solve	5
Defining a Simulation in ANSYS CFX-Pre	6
Obtaining a Solution Using ANSYS CFX-Solver Manager	12
Viewing the Results in ANSYS CFX-Post	15

Tutorial 1a: Simulating Flow in a Static Mixer Using Workbench

Introduction	31
Before You Begin	32
Tutorial 1a Features	32

Table of Contents: Tutorial 2: Flow in a Static Mixer (Refined Mesh)

Overview of the Problem to Solve.....	33
Defining a Simulation in ANSYS CFX-Pre	34
Obtaining a Solution Using ANSYS CFX-Solver Manager	41
Viewing the Results in ANSYS CFX-Post	43

**Tutorial 2:
Flow in a Static Mixer
(Refined Mesh)**

Introduction	59
Tutorial 2 Features	60
Overview of the Problem to Solve.....	60
Defining a Simulation using General Mode in ANSYS CFX-Pre	61
Obtaining a Solution Using Interpolation with ANSYS CFX-Solver	66
Viewing the Results in ANSYS CFX-Post	68

**Tutorial 3:
Flow in a Process Injection Mixing Pipe**

Introduction	77
Tutorial 3 Features	78
Overview of the Problem to Solve.....	78
Defining a Simulation using General Mode in ANSYS CFX-Pre	79
Obtaining a Solution Using ANSYS CFX-Solver Manager	87
Viewing the Results in ANSYS CFX-Post	88

**Tutorial 4:
Flow from a Circular Vent**

Introduction	93
Tutorial 4 Features	94
Overview of the Problem to Solve.....	95
Defining a Steady-State Simulation in ANSYS CFX-Pre	95
Obtaining a Solution to the Steady-State Problem.....	99
Defining a Transient Simulation in ANSYS CFX-Pre.....	100
Obtaining a Solution to the Transient Problem	104
Viewing the Results in ANSYS CFX-Post	105

**Tutorial 5:
Flow Around a Blunt Body**

Introduction	109
Tutorial 5 Features	109
Overview of the Problem to Solve.....	111
Defining a Simulation in ANSYS CFX-Pre	111

Obtaining a Solution Using ANSYS CFX-Solver Manager	116
Viewing the Results in ANSYS CFX-Post	119

**Tutorial 6:
Buoyant Flow in a Partitioned Cavity**

Introduction	127
Tutorial 6 Features	128
Overview of the Problem to Solve.....	128
Defining a Simulation in ANSYS CFX-Pre	129
Obtaining a Solution using ANSYS CFX-Solver Manager	134
Viewing the Results in ANSYS CFX-Post	135

**Tutorial 7:
Free Surface Flow Over a Bump**

Introduction	139
Tutorial 7 Features	139
Overview of the Problem to Solve.....	140
Defining a Simulation in ANSYS CFX-Pre	141
Obtaining a Solution using ANSYS CFX-Solver Manager	148
Viewing the Results in ANSYS CFX-Post	149
Using a Supercritical Outlet Condition	154

**Tutorial 8:
Supersonic Flow Over a Wing**

Introduction	155
Tutorial 8 Features	155
Overview of the Problem to Solve.....	157
Defining a Simulation in ANSYS CFX-Pre	157
Obtaining a Solution using ANSYS CFX-Solver Manager	162
Viewing the Results in ANSYS CFX-Post	162

**Tutorial 9:
Flow Through a Butterfly Valve**

Introduction	165
Tutorial 9 Features	165
Overview of the Problem to Solve.....	166
Defining a Simulation in ANSYS CFX-Pre	167
Obtaining a Solution using ANSYS CFX-Solver Manager	180
Viewing the Results in ANSYS CFX-Post	180

Tutorial 10:

Flow in a Catalytic Converter

Introduction	185
Tutorial 10 Features	185
Overview of the Problem to Solve.....	186
Defining a Simulation in ANSYS CFX-Pre	187
Obtaining a Solution using ANSYS CFX-Solver Manager	193
Viewing the Results in ANSYS CFX-Post	194

**Tutorial 11:
Non-Newtonian Fluid Flow in an Annulus**

Introduction	199
Tutorial 11 Features	200
Overview of the Problem to Solve.....	201
Defining a Simulation in ANSYS CFX-Pre	201
Obtaining a Solution using ANSYS CFX-Solver Manager	205
Viewing the Results in ANSYS CFX-Post	206

**Tutorial 12:
Flow in an Axial Rotor/Stator**

Introduction	207
Tutorial 12 Features	208
Overview of the Problem to Solve.....	209
Defining a Frozen Rotor Simulation in ANSYS CFX-Pre	210
Obtaining a Solution to the Frozen Rotor Model.....	214
Viewing the Frozen Rotor Results in ANSYS CFX-Post	215
Setting up a Transient Rotor-Stator Calculation.....	216
Obtaining a Solution to the Transient Rotor-Stator Model.....	219
Viewing the Transient Rotor-Stator Results in ANSYS CFX-Post	220

**Tutorial 13:
Reacting Flow in a Mixing Tube**

Introduction	223
Tutorial 13 Features	223
Overview of the Problem to Solve.....	224
Outline of the Process	224
Defining a Simulation in ANSYS CFX-Pre	225
Obtaining a Solution using ANSYS CFX-Solver Manager	237
Viewing the Results in ANSYS CFX-Post	237

Tutorial 14:

Conjugate Heat Transfer in a Heating Coil

Introduction	239
Tutorial 14 Features	240
Overview of the Problem to Solve.....	241
Defining a Simulation in ANSYS CFX-Pre	241
Obtaining a Solution using ANSYS CFX-Solver Manager	246
Viewing the Results in ANSYS CFX-Post	246
Exporting the Results to ANSYS	247

**Tutorial 15:
Multiphase Flow in Mixing Vessel**

Introduction	251
Tutorial 15 Features	252
Overview of the Problem to Solve.....	253
Defining a Simulation in ANSYS CFX-Pre	253
Obtaining a Solution using ANSYS CFX-Solver Manager	265
Viewing the Results in ANSYS CFX-Post	265

**Tutorial 16:
Gas-Liquid Flow in an Airlift Reactor**

Introduction	269
Tutorial 16 Features	270
Overview of the Problem to Solve.....	270
Defining a Simulation in ANSYS CFX-Pre	271
Obtaining a Solution using ANSYS CFX-Solver Manager	277
Viewing the Results in ANSYS CFX-Post	278
Additional Fine Mesh Simulation Results	280

**Tutorial 17:
Air Conditioning Simulation**

Introduction	283
Tutorial 17 Features	284
Overview of the Problem to Solve.....	285
Defining a Simulation in ANSYS CFX-Pre	285
Obtaining a Solution using ANSYS CFX-Solver Manager	295
Viewing the Results in ANSYS CFX-Post	295

**Tutorial 18:
Combustion and Radiation in a Can Combustor**

Introduction	299
--------------------	-----

Table of Contents: Tutorial 19: Cavitation Around a Hydrofoil

Tutorial 18 Features	300
Overview of the Problem to Solve.....	301
Using Eddy Dissipation and P1 Models	301
Defining a Simulation in ANSYS CFX-Pre	302
Obtaining a Solution using ANSYS CFX-Solver Manager	307
Viewing the Results in ANSYS CFX-Post	308
Laminar Flamelet and Discrete Transfer Models	311
Further Postprocessing	316

**Tutorial 19:
Cavitation Around a Hydrofoil**

Introduction	317
Tutorial 19 Features	318
Overview of the Problem to Solve.....	319
Creating an Initial Simulation	319
Obtaining an Initial Solution using ANSYS CFX-Solver Manager	323
Viewing the Results of the Initial Simulation	324
Preparing a Simulation with Cavitation.....	326
Obtaining a Cavitation Solution using ANSYS CFX-Solver Manager	328
Viewing the Results of the Cavitation Simulation	328

**Tutorial 20:
Fluid Structure Interaction and Mesh Deformation**

Introduction	331
Tutorial 20 Features	332
Overview of the Problem to Solve.....	333
Using CEL Expressions to Govern Mesh Deformation	334
Using a Junction Box Routine to Govern Mesh Deformation	343

**Tutorial 21:
Oscillating Plate with Two-Way Fluid-Structure Interaction**

Introduction	353
Tutorial 21 Features	354
Overview of the Problem to Solve.....	354
Setting up the Solid Physics in Simulation (ANSYS Workbench)	355
Setting up the Fluid Physics and ANSYS Multi-field Settings in ANSYS CFX-Pre.....	358
Obtaining a Solution using ANSYS CFX-Solver Manager	364
Viewing Results in ANSYS CFX-Post	365

Tutorial 22:

Optimizing Flow in a Static Mixer

Introduction	369
Tutorial 22 Features	370
Overview of the Problem to Solve.....	370
Creating the Project.....	371
Creating the Geometry	372
Creating the Mesh.....	377
Overview of ANSYS CFX-Pre	380
Setting the Output Parameter in ANSYS CFX-Post	386
Running Design Studies in DesignXplorer.....	387

Tutorial 23: Aerodynamic & Structural Performance of a Centrifugal Compressor

Introduction	395
Tutorial 23 Features	396
Overview of the Problem to Solve.....	397
Reviewing the Centrifugal Compressor Design	397
Creating the Mesh in ANSYS TurboGrid	398
Defining the Aerodynamic Simulation in ANSYS CFX-Pre	401
Obtaining a Solution using ANSYS CFX-Solver Manager	403
Viewing the Results in ANSYS CFX-Post	404
Importing Geometry into DesignModeler	405
Simulating Structural Stresses Due to Pressure Loads.....	406
Simulating Structural Stresses Due to Rotation	407

Introduction to the ANSYS CFX Tutorials

Overview

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

Earlier tutorials introduce general principles used in ANSYS CFX, including setting up the physical models, running ANSYS CFX-Solver and visualizing the results. The remaining tutorials highlight specialized features of ANSYS CFX.

Files required to complete each tutorial is listed in the introduction to the tutorial, and located in `<CFXROOT>/examples`, where `<CFXROOT>` is the installation directory.

Setting the Working Directory

One of the first things you must do when using ANSYS CFX is to set a working directory. The working directory is the default location for loading and saving files for a particular session or project.

The working directory is set according to how you run ANSYS CFX:

- **Workbench**
Set the working directory by saving a project file.
- **Standalone**
Set the working directory by entering it in CFX Launcher.

Changing the Display Colors

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

- In standalone mode (i.e., after using CFX Launcher to launch ANSYS CFX-Pre or ANSYS CFX-Post):
 - a. Select **Edit > Options**.
The **Options** dialog box appears.
 - b. Adjust the color settings under `CFX-Pre > Viewer` (for ANSYS CFX-Pre) or `CFX-Post > Viewer` (for ANSYS CFX-Post).
 - c. Click **OK**.
- In ANSYS Workbench:
 - a. Select **Tools > Options** from the Project page.
 - b. Adjust the color settings under `Common Settings > Graphics Style`.
 - c. Click **OK**.

Tutorial 1: Simulating Flow in a Static Mixer Using CFX in Standalone Mode

Introduction

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.

This tutorial includes:

- [Before You Begin](#) (p. 4)
- [Tutorial 1 Features](#) (p. 4)
- [Overview of the Problem to Solve](#) (p. 5)
- [Defining a Simulation in ANSYS CFX-Pre](#) (p. 6)
- [Obtaining a Solution Using ANSYS CFX-Solver Manager](#) (p. 12)
- [Viewing the Results in ANSYS CFX-Post](#) (p. 15)

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

- [Setting the Working Directory](#) (p. 1)
- [Changing the Display Colors](#) (p. 2)

To learn how to perform these tasks in Workbench, see [Tutorial 1a: Simulating Flow in a Static Mixer Using Workbench](#) (p. 31 in "ANSYS CFX Tutorials").

Before You Begin

Create a working directory for your files. Once this is done, copy the sample files used in this tutorial to your working directory from the installation folder for your software (<CFXROOT>/examples/ (for example, C:\Program Files\ANSYS Inc\v110\CFX\examples)) to avoid overwriting source files provided with your installation. If you plan to use a session file, please refer to [Playing a Session File](#) (p. 7).

Sample files used by this tutorial are:

- `StaticMixerMesh.gtm`
- `StaticMixer.pre`

Tutorial 1 Features

This tutorial addresses the following features of ANSYS CFX.

Component	Feature	Details	
ANSYS CFX-Pre	User Mode	Quick Setup Wizard	
	Simulation 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	
ANSYS CFX-Post	Plots	Animation	
		Contour	
		Outline Plot (Wireframe)	
		Point	
		Slice Plane Streamline	

In this tutorial you will learn about:

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

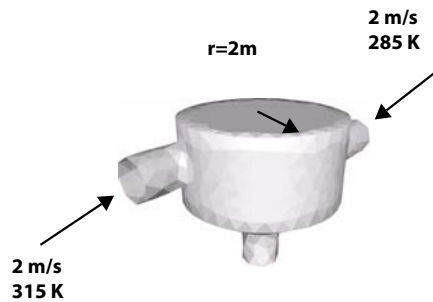
Overview of the Problem to Solve

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.

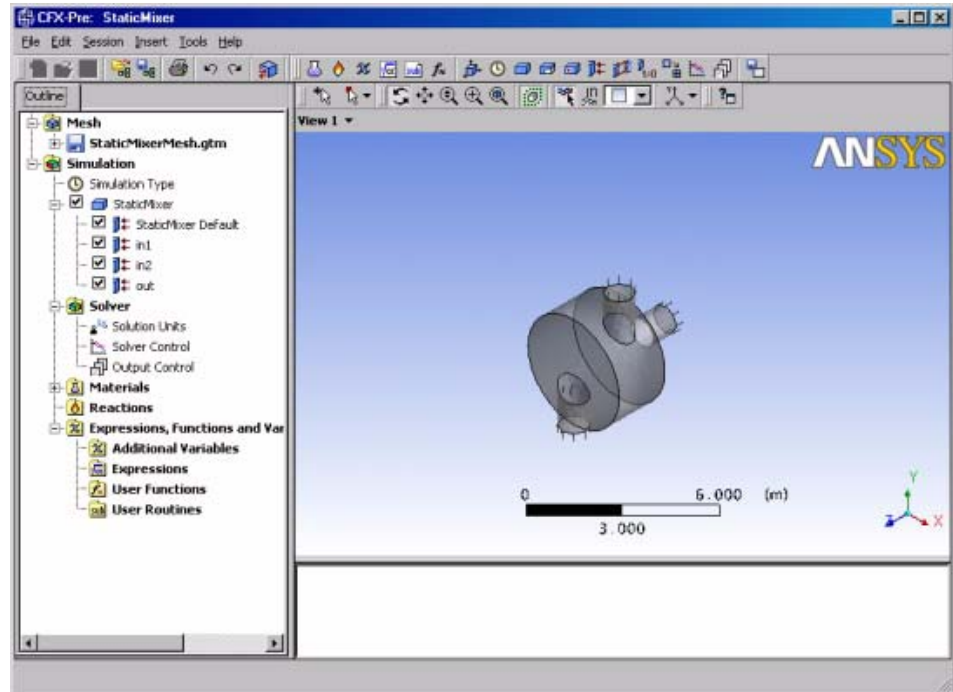
Your goal in this tutorial is to understand how to use ANSYS CFX to determine the speed and temperature of the water when it exits the static mixer.

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



Defining a Simulation in ANSYS CFX-Pre

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



In the image above, the left pane of ANSYS CFX-Pre displays the **Outline**. When you double-click on 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).

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 ANSYS CFX-Pre.

Workflow Overview

This tutorial follows the general workflow for Quick Setup mode:

1. [Creating a New Simulation](#) (p. 7)
2. [Setting the Physics Definition](#) (p. 7)
3. [Importing a Mesh](#) (p. 7)
4. [Defining Model Data](#) (p. 9)
5. [Defining Boundaries](#) (p. 9)

6. [Setting Boundary Data](#) (p. 9)
7. [Setting Flow Specification](#) (p. 9)
8. [Setting Temperature Specification](#) (p. 10)
9. [Reviewing the Boundary Condition Definitions](#) (p. 10)
10. [Creating the Second Inlet Boundary Definition](#) (p. 10)
11. [Creating the Outlet Boundary Definition](#) (p. 10)
12. [Moving to General Mode](#) (p. 11)
13. [Writing the Solver \(.def\) File](#) (p. 11)

Playing a Session File

If you want to skip past these instructions and have ANSYS CFX-Pre set up the simulation automatically, you can select **Session > Play Tutorial** from the menu in ANSYS CFX-Pre, then run the appropriate session file. For details, see [Playing the Session File and Starting ANSYS CFX-Solver Manager](#) (p. 12). After you have played the session file, proceed to [Obtaining a Solution Using ANSYS CFX-Solver Manager](#) (p. 12).

Creating a New Simulation

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

Procedure

1. If required, launch ANSYS CFX-Pre.
2. Select **File > New Simulation**.
The **New Simulation File** dialog box is displayed.
3. 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**.

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

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.


Procedure

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

Importing a Mesh

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

Procedure

1. In **Simulation Definition**, under **Mesh File**, click *Browse* .
The **Import Mesh** dialog box appears.



2. Under **File type**, select `CFX Mesh (*.gtm *.cfx)`.
3. From your working directory, select `StaticMixerMesh.gtm`.
4. Click **Open**.
The mesh loads.
5. Click **Next**.

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.


Using the Zoom Tools

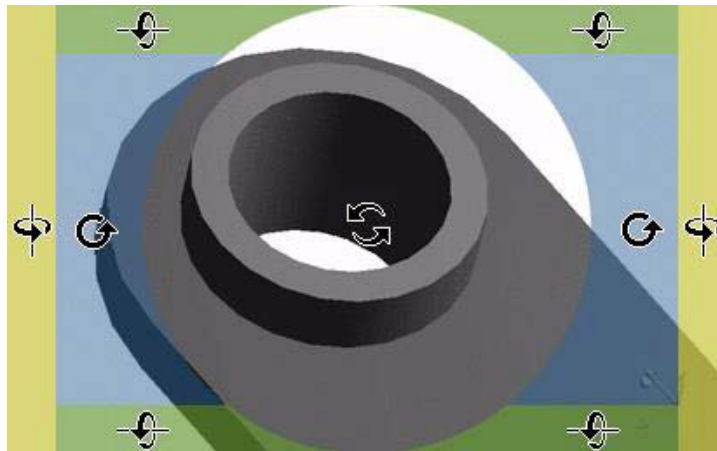
There are several icons available for controlling the level of zoom in the viewer.

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.

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 Towards-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.

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.

Procedure

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**.

Defining Boundaries

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

Procedure

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

Setting Boundary Data

Once boundaries are created, you need to create associated data. Based on [Figure 1](#), you will define the first inlet boundary condition's velocity and temperature.

Procedure

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

Setting Flow Specification

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

Procedure

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

Setting Temperature Specification

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

Procedure

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

Reviewing the Boundary Condition Definitions

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

Based on [Figure 1](#), 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.

Procedure

1. 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]

Creating the Second Inlet Boundary Definition

Based on [Figure 1](#), 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.

Procedure

1. Under **Boundary Definition**, right-click in the selector area and select **New**.
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]

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**.

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.

Procedure

1. 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.


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.

Procedure

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**.

Writing the Solver (.def) File

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

Procedure

1. Click **Write Solver File** .

The **Write Solver File** dialog box is displayed.


2. Set **File name** to `StaticMixer.def`.
3. Ensure that `Start Solver Manager` is selected from the drop down menu located in the top-right corner of the dialog box.
4. Select **Quit ANSYS CFX-Pre**.
This forces standalone ANSYS CFX-Pre to close after the definition file has been written.
5. Click **Save**.
6. If you are notified the file already exists, click **Overwrite**.
This file is provided in the tutorial directory and may exist in your tutorial folder if you have copied it there.
7. If prompted, click **Yes** or **Save & Quit** to save `StaticMixer.cfx`.
The definition file (`StaticMixer.def`) and the simulation file (`StaticMixer.cfx`) are created. ANSYS CFX-Solver Manager automatically starts and the definition file is set in the **Define Run** dialog box.
8. Proceed to [Obtaining a Solution Using ANSYS CFX-Solver Manager](#) (p. 12).

Playing the Session File and Starting ANSYS 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 ANSYS CFX-Solver Manager](#) (p. 12).

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

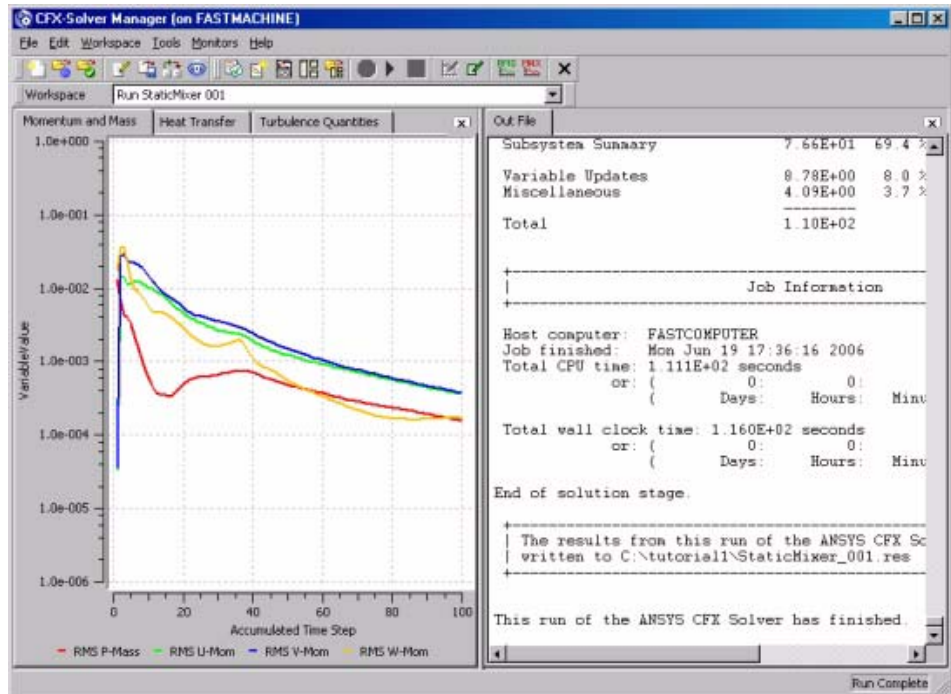
Procedure

1. If required, launch ANSYS CFX-Pre.
2. Select **Session > Play Tutorial**.
3. Select `StaticMixer.pre`.
4. Click **Open**.
A definition file is written.
5. Select **File > Quit**.
6. Launch the ANSYS CFX-Solver Manager from CFX Launcher.
7. After the ANSYS CFX-Solver starts, select **File > Define Run**.
8. Under **Definition File**, click *Browse* .
9. Select `StaticMixer.def`, located in the working directory.
10. Proceed to [Obtaining a Solution Using ANSYS CFX-Solver Manager](#) (p. 12).

Obtaining a Solution Using ANSYS CFX-Solver Manager

ANSYS 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 ANSYS 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 ANSYS CFX-Solver Manager window (also adjustable).



One window shows the convergence history plots and the other displays text output from ANSYS 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**).

Start the Run

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

When ANSYS CFX-Solver Manager is launched automatically from ANSYS 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 ANSYS CFX-Solver without having to define settings and values.

Procedure

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

Tutorial 1: Simulating Flow in a Static Mixer Using CFX in Standalone Mode: Obtaining a Solution Using ANSYS

ANSYS 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 ANSYS CFX-Solver Manager operates.

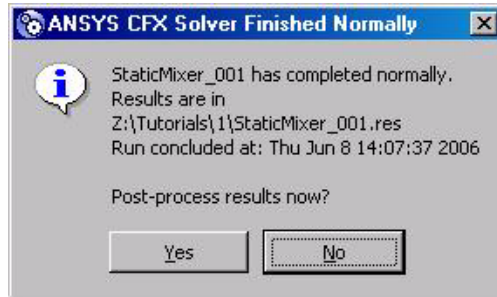
Note: Once the second iteration appears, data begins to plot. Plotting may take a long time depending on the amount of data to process. Let the process run.

Move from ANSYS CFX-Solver to ANSYS CFX-Post

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

Procedure

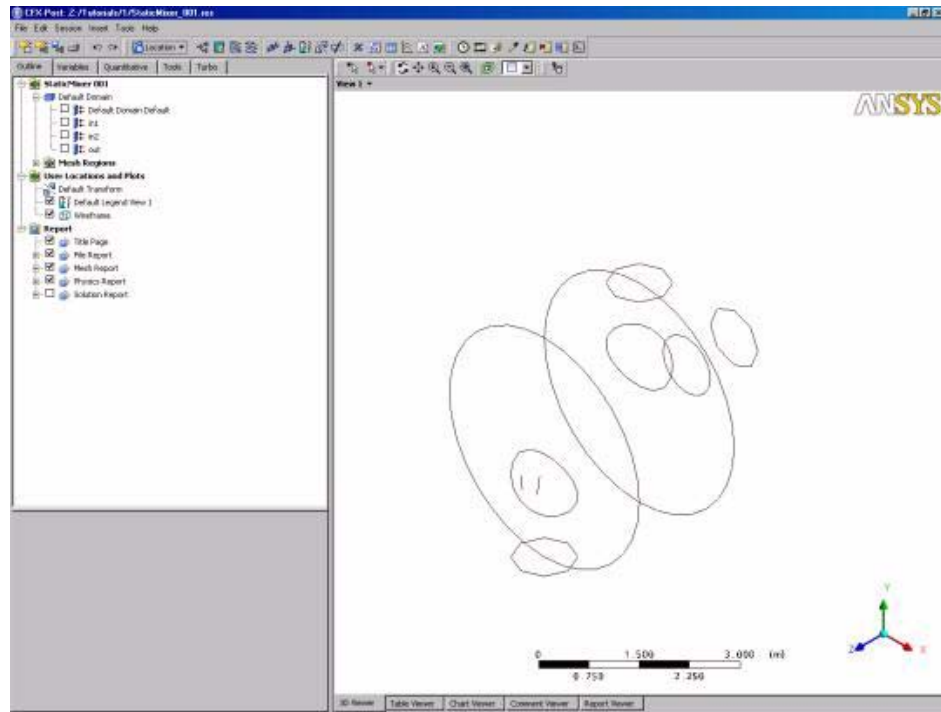
1. When ANSYS CFX-Solver is finished, click **Yes** to post-process the results.



After a short pause, ANSYS CFX-Post starts and ANSYS CFX-Solver Manager closes.

Viewing the Results in ANSYS CFX-Post

When ANSYS CFX-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 ANSYS CFX-Pre.

Workflow Overview

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

1. [Setting the Edge Angle for a Wireframe Object](#) (p. 16)
2. [Creating a Point for the Origin of the Streamline](#) (p. 17)
3. [Creating a Streamline Originating from a Point](#) (p. 18)
4. [Rearranging the Point](#) (p. 19)
5. [Configuring a Default Legend](#) (p. 19)
6. [Creating a Slice Plane](#) (p. 20)
7. [Defining Slice Plane Geometry](#) (p. 21)
8. [Configuring Slice Plane Views](#) (p. 21)
9. [Rendering Slice Planes](#) (p. 22)
10. [Coloring the Slice Plane](#) (p. 23)
11. [Moving the Slice Plane](#) (p. 23)
12. [Adding Contours](#) (p. 24)
13. [Working with Animations](#) (p. 25)
14. [Showing the Animation Dialog Box](#) (p. 25)

15. [Creating the First Keyframe](#) (p. 26)
16. [Creating the Second Keyframe](#) (p. 26)
17. [Viewing the Animation](#) (p. 27)
18. [Modifying the Animation](#) (p. 28)
19. [Saving to MPEG](#) (p. 29)

Setting the Edge Angle for a Wireframe Object

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

By default, ANSYS CFX-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** control the visibility of each object. Currently only the `Wireframe` and `Default Legend` objects have visibility selected. 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.

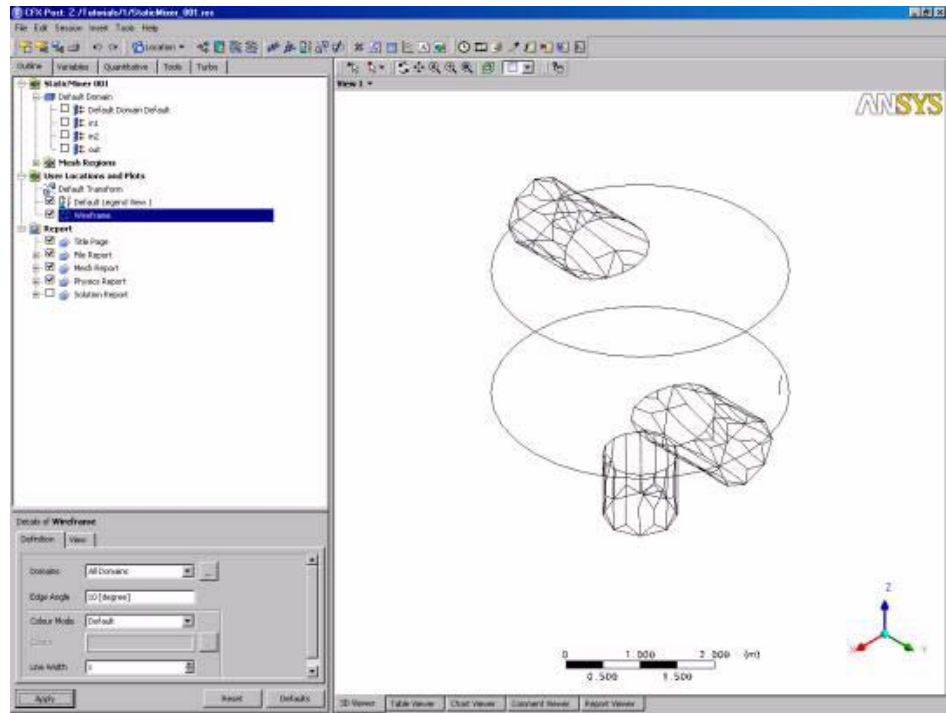
For this geometry, a setting of approximately 15° lets you view the model location without displaying an excessive amount of the surface mesh.

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

Procedure

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 on 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.
10. Right-click on a blank area anywhere in the viewer, select **Predefined Camera** from the shortcut menu and select **View Towards -X**.

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.

Procedure

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.

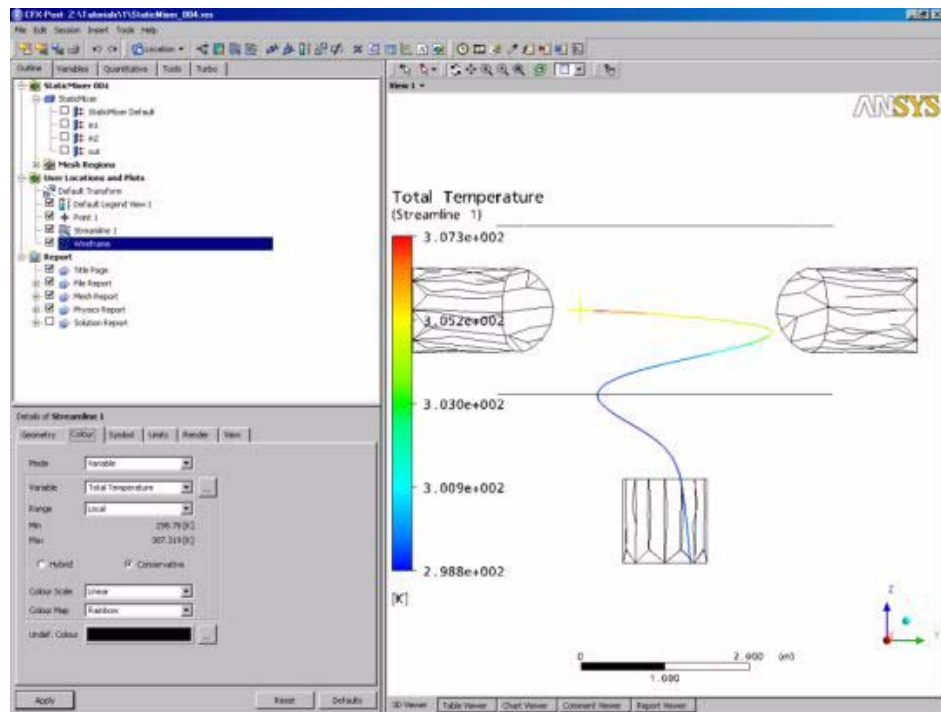
Creating a Streamline Originating from a Point

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

Procedure


- From the main menu, select **Insert > Streamline**.
You can also use the toolbars to create a variety of objects. Later modules and tutorials will explore this further.
- Click **OK**.
This accepts the default name.
- Under **Definition**, in **Start From**, ensure that **Point 1** is set.
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.
- Click the **Color** tab.
- Set **Mode** to **Variable**.
- Set **Variable** to **Total Temperature**.
- Set **Range** to **Local**.
- 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.



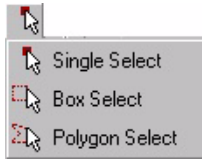
Rearranging the Point

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 Towards -X**) and by using **Zoom Box**  if you prefer to change the display.

Procedure

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 selection tools, click **Single Select**.



While in this mode, the normal behavior of the left mouse button is disabled.


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 Towards -X**.

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 plot¹.

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.

The default legend displays values for the last eligible plot that was opened in the details view. To maintain a legend definition during an ANSYS CFX-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**.

Procedure

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.

Creating a Slice Plane

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

In ANSYS CFX-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.

Procedure

1. From the main menu, select **Insert > Location > Plane** or click **Location > Plane**.
2. In the **New Plane** window, type: `Slice`
3. Click **OK**.
The **Geometry, Color, Render** and **View** tabs let you switch between settings.
4. Click the **Geometry** tab.

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).

Procedure

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

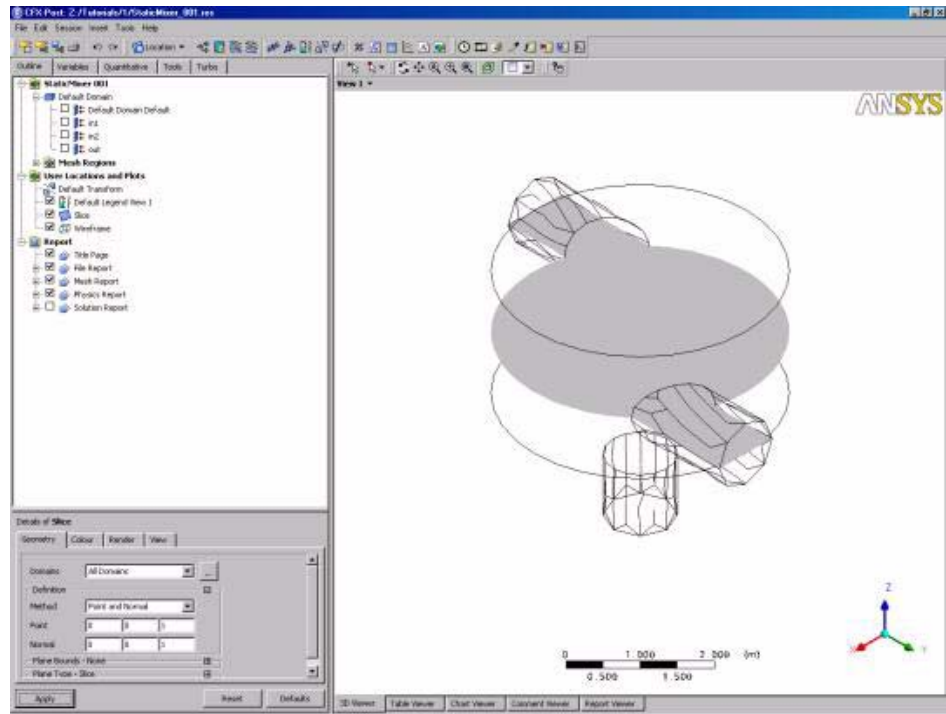
Configuring Slice Plane Views



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

Procedure

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.

Rendering Slice Planes

Render settings determine how the plane is drawn.

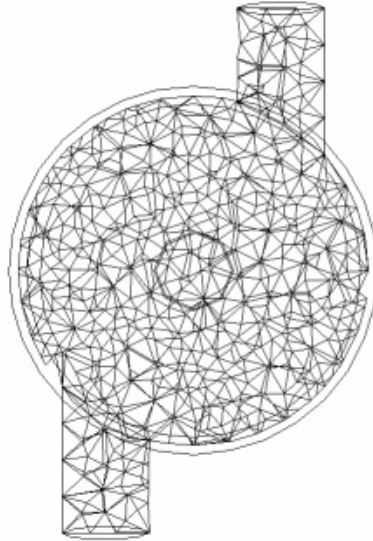
Procedure

1. Select the **Render** tab.
2. Clear **Draw Faces**.
3. Select **Draw Lines**.
4. Under **Draw 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 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 Towards -Z**.

The image shown below can be used for comparison with tutorial 2 (in the section [Creating a Slice Plane](#) (p. 68)), where a refined mesh is used.



Coloring the Slice Plane

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

Procedure

1. Apply the following settings to `slice`



Tab	Setting	Value
Color	Mode	Variable*
	Variable	Temperature
Render	Draw Faces	(Selected)
	Draw Lines	(Cleared)

*. 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.


Moving the Slice Plane

The plane can be moved to different locations.

- Procedure**
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 for *slice* by clearing the check box next to *slice* in the **Outline**.

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).

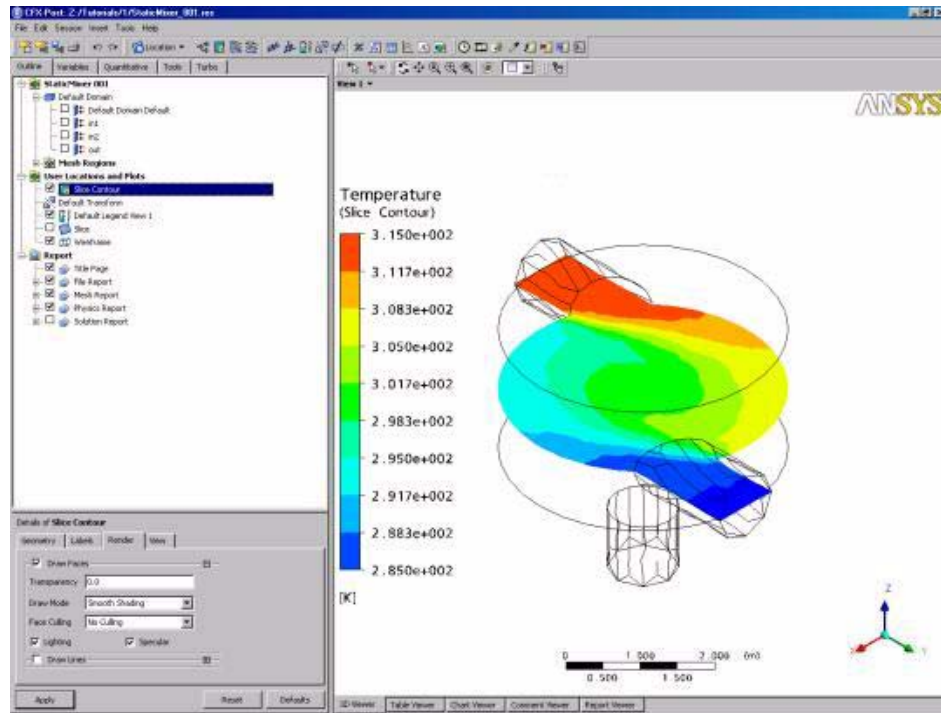
- Procedure**
1. Select **Insert > Contour** from the main menu or click *Contour* .
 - The **New Contour** dialog box is displayed.
 2. Set **Name** to *slice Contour*.
 3. Click **OK**.
 4. Apply the following settings

Tab	Setting	Value
Geometry	Locations	Slice
	Variable	Temperature
Render	Draw Faces	(Selected)

5. Click **Apply**.

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

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



Note: Make sure that the checkbox next to **Slice** in the **Outline** is cleared.

Working with Animations

Animations build transitions between views for development of video files.

Workflow Overview

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

1. [Showing the Animation Dialog Box](#) (p. 25)
2. [Creating the First Keyframe](#) (p. 26)
3. [Creating the Second Keyframe](#) (p. 26)
4. [Viewing the Animation](#) (p. 27)
5. [Modifying the Animation](#) (p. 28)
6. [Saving to MPEG](#) (p. 29)

Showing the Animation Dialog Box

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

Procedure


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

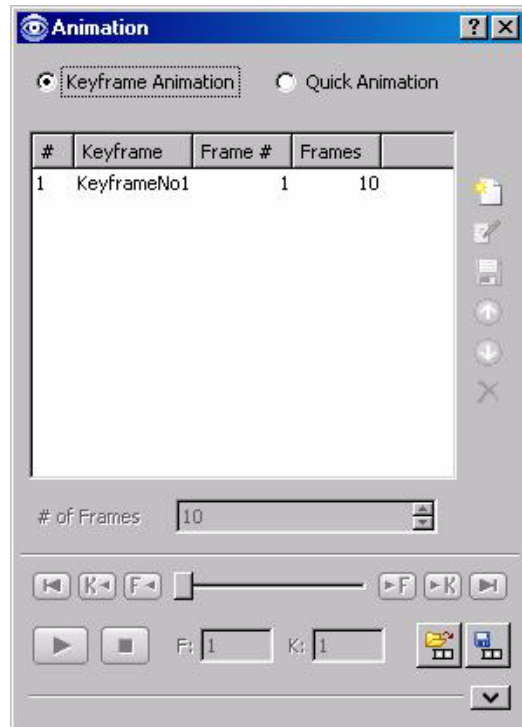
The **Animation** dialog box can be repositioned as required.

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.

Procedure


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*, clear the visibility of *Slice Contour* and select 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.



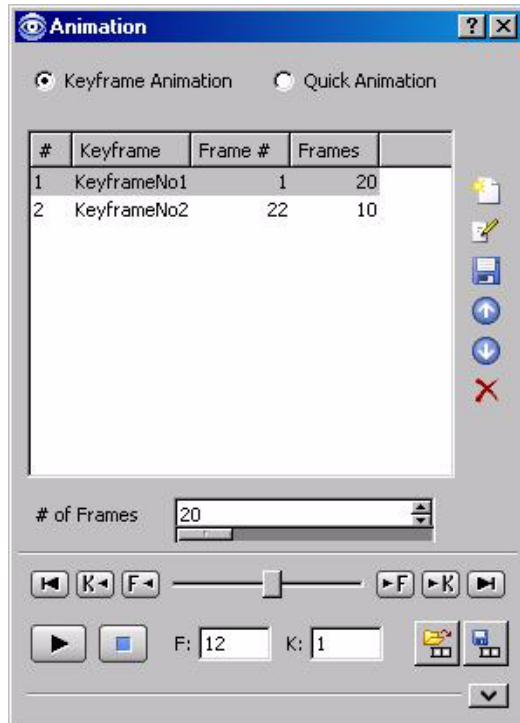
Creating the Second Keyframe

Keyframes are required in order to produce an animation.

Procedure

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`.




Viewing the Animation

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

Synopsis

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 keyframes listed beside the last keyframe is ignored.

Procedure

1. 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.

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.



Procedure

1. Apply the following settings to `Slice`


Tab	Setting	Value
Geometry	Point	0, 0, 1.99
Color	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	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`.

Saving to MPEG

By defining the geometry and then saving to MPEG, the results can be saved to a video file.

Procedure

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. Click **Save MPEG**.
3. Click *Browse*  next to **Save MPEG**.
4. Under **File name** type: `StaticMixer.mpg`
5. If required, set the path location to a different folder.
6. Click **Save**.
The MPEG file name (including path) is set. At this point, the animation has not yet been produced.
7. Click *Previous Keyframe* .
Wait a moment as the display updates the keyframe display.
8. Click *Play the animation* .
9. If prompted to overwrite an existing movie click **Overwrite**.
The animation plays and builds an MPEG file.
10. 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.
11. Click **Cancel** to close the dialog box.
12. Close the **Animation** dialog box.
13. Review the animation in third-party software as required.

Exiting ANSYS CFX-Post

When finished with ANSYS CFX-Post exit the current window:

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

Tutorial 1a: Simulating Flow in a Static Mixer Using Workbench

Introduction

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.

This tutorial comprises:

- [Before You Begin](#) (p. 32)
- [Tutorial 1a Features](#) (p. 32)
- [Overview of the Problem to Solve](#) (p. 33)
- [Defining a Simulation in ANSYS CFX-Pre](#) (p. 34)
- [Obtaining a Solution Using ANSYS CFX-Solver Manager](#) (p. 41)
- [Viewing the Results in ANSYS CFX-Post](#) (p. 43)

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

- [Setting the Working Directory](#) (p. 1)
- [Changing the Display Colors](#) (p. 2)

To learn how to perform these tasks using CFX in Standalone mode, see [Tutorial 1: Simulating Flow in a Static Mixer Using CFX in Standalone Mode](#) (p. 3 in "ANSYS CFX Tutorials").

Before You Begin

Create a working directory for your files. Once this is done, copy the sample files used in this tutorial to your working directory from the installation folder for your software (<CFXROOT>/examples/ (for example, C:\Program Files\ANSYS Inc\v110\CFX\examples)) to avoid overwriting source files provided with your installation. If you plan to use a session file, please refer to [Playing a Session File](#) (p. 35).

Sample files used by this tutorial are:

- `StaticMixerMesh.gtm`
- `StaticMixer.pre`

Tutorial 1a Features

This tutorial addresses the following features of ANSYS CFX.

Component	Feature	Details	
ANSYS CFX-Pre	User Mode	Quick Setup Wizard	
	Simulation 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: Adiabatic			
	Timestep	Physical Time Scale	
ANSYS CFX-Post	Plots	Animation	
		Contour	
		Outline Plot (Wireframe)	
		Point	
		Slice Plane	
		Streamline	

In this tutorial you will learn about:

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

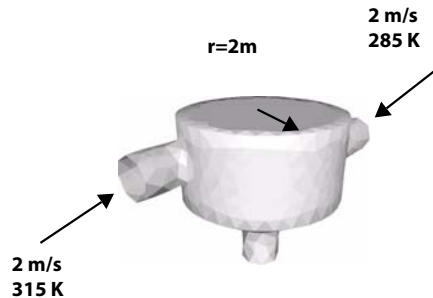
Overview of the Problem to Solve

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.

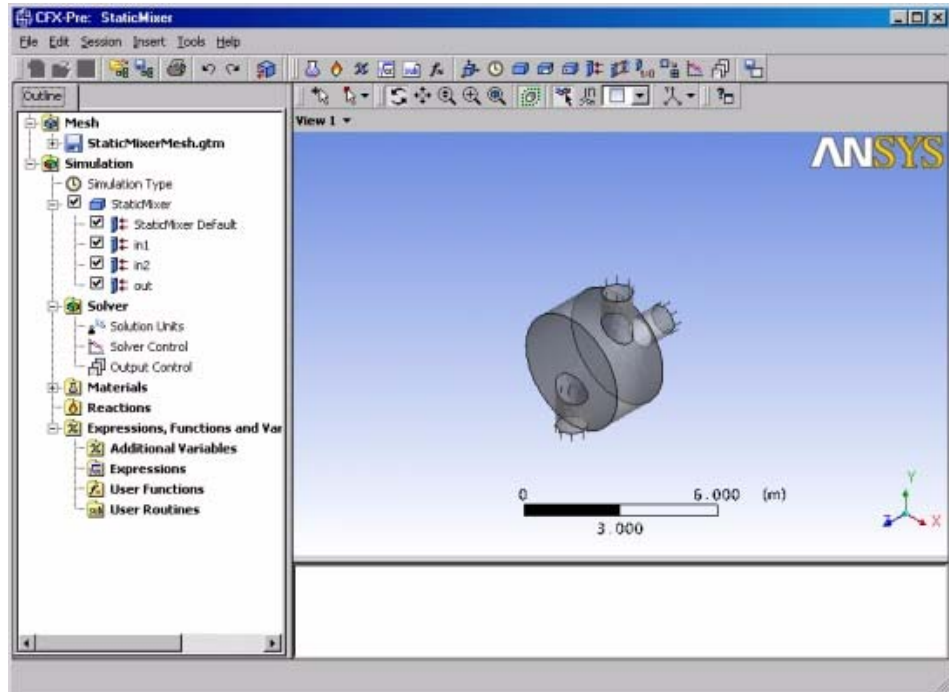
Your goal in this tutorial is to understand how to use ANSYS CFX to determine the speed and temperature of the water when it exits the static mixer.

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



Defining a Simulation in ANSYS CFX-Pre

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



In the image above, the left pane of ANSYS CFX-Pre displays the **Outline**. When you double-click on 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).

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 ANSYS CFX-Pre.

Workflow Overview

This tutorial follows the general workflow for Quick Setup mode:

1. [Creating a New Simulation](#) (p. 35)
2. [Setting the Physics Definition](#) (p. 35)
3. [Importing a Mesh](#) (p. 36)
4. [Defining Model Data](#) (p. 37)
5. [Defining Boundaries](#) (p. 37)

6. [Setting Boundary Data](#) (p. 37)
7. [Setting Flow Specification](#) (p. 37)
8. [Setting Temperature Specification](#) (p. 38)
9. [Reviewing the Boundary Condition Definitions](#) (p. 38)
10. [Creating the Second Inlet Boundary Definition](#) (p. 38)
11. [Creating the Outlet Boundary Definition](#) (p. 39)
12. [Moving to General Mode](#) (p. 39)
13. [Writing the Solver \(.def\) File](#) (p. 40)


Playing a Session File

If you want to skip past these instructions and have ANSYS CFX-Pre set up the simulation automatically, you can select **Session > Play Tutorial** from the menu in ANSYS CFX-Pre, then run the appropriate session file. For details, see [Playing the Session File and Starting ANSYS CFX-Solver Manager](#) (p. 40). After you have played the session file, proceed to [Obtaining a Solution Using ANSYS CFX-Solver Manager](#) (p. 41).

Creating a New Simulation

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

Procedure

1. If required, launch ANSYS Workbench.
2. Click **Empty Project**.
The Project page appears displaying an unsaved project.
3. Select **File > Save** or click *Save* .
4. If required, set the path location to the working folder you created for this tutorial.
5. Under **File name**, type: `StaticMixer`
6. Click **Save**.
7. On the left-hand task bar under **Advanced CFD**, click **Start CFX-Pre**.
8. Select **File > New Simulation**.
9. Select **Quick Setup** in the **New Simulation File** dialog box and click **OK**.
10. Select **File > Save Simulation As**.
11. Under **File name**, type: `StaticMixer`
12. Click **Save**.

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.


Procedure

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

Importing a Mesh

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

Procedure



1. In **Simulation Definition**, under **Mesh File**, click *Browse*  .
The **Import Mesh** dialog box appears.
2. Under **File type**, select *CFX Mesh (*.gtm)*.
3. From your working directory, select *StaticMixerMesh.gtm*.
4. Click **Open**.
The mesh loads.
5. Click **Next**.

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.


Using the Zoom Tools

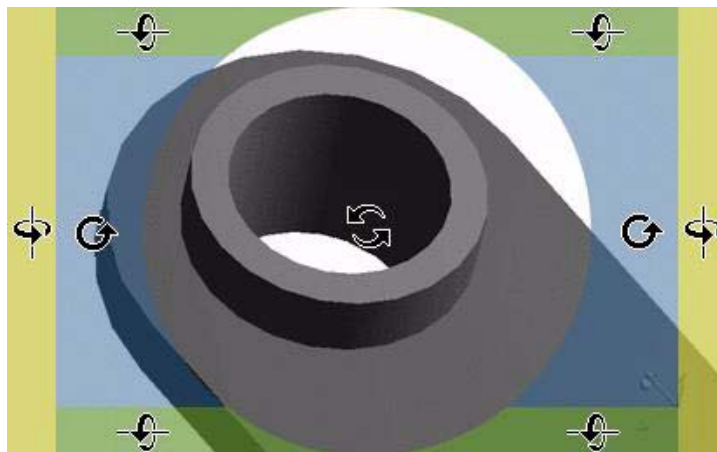
There are several icons available for controlling the level of zoom in the viewer.

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.

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 Towards-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.

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 modelled 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.

Procedure

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**.

Defining Boundaries

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

Procedure

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

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

Setting Boundary Data

Once boundaries are created, you need to create associated data. Based on [Figure 1](#), you will define the first inlet boundary condition to have a velocity of 2 m/s and a temperature of 315 K at one of the side inlets.

Procedure

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

Setting Flow Specification

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

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

Setting Temperature Specification

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

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

Reviewing the Boundary Condition Definitions

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

Based on [Figure 1](#), 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.

- Procedure**
1. 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]

Creating the Second Inlet Boundary Definition

Based on [Figure 1](#), 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.

- Procedure**
1. Under **Boundary Definition**, right-click in the selector area and select **New**.
 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]

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**.

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.

Procedure

1. 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.


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.


Procedure

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**.

Writing the Solver (.def) File

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

Procedure



1. Click *Write Solver File* .
The **Write Solver File** dialog box is displayed.
2. Set **File name** to `StaticMixer.def`.
3. Ensure that `Start Solver Manager` is selected from the drop down menu located in the top-right corner of the dialog box.
4. Click **Save**.
5. If you are notified the file already exists, click **Overwrite**.
This file is provided in the tutorial directory and may exist in your tutorial folder if you have copied it there.
6. If prompted, click **Yes** or **Save & Quit** to save `StaticMixer.cfx`.
The definition file (`StaticMixer.def`) and the simulation file (`StaticMixer.cfx`) are created. ANSYS CFX-Solver Manager automatically starts and the definition file is set in the **Define Run** dialog box.
7. Proceed to [Obtaining a Solution Using ANSYS CFX-Solver Manager](#) (p. 41).

Playing the Session File and Starting ANSYS 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 ANSYS CFX-Solver Manager](#) (p. 41).

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

Procedure

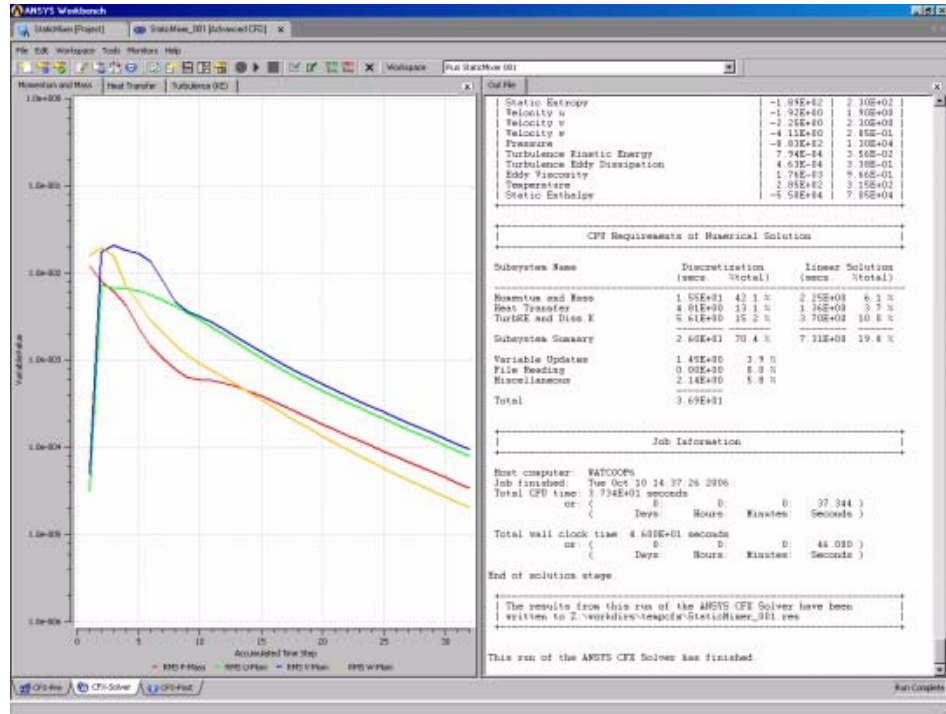
1. If required, launch ANSYS Workbench.
2. Click **Empty Project**.
3. Select **File > Save** or click *Save* .
4. Under **File name**, type: `StaticMixer`
5. Click **Save**.
6. Click **Start CFX-Pre**.
7. Select **Session > Play Tutorial**.
8. Select `StaticMixer.pre`.
9. Click **Open**.
A definition file is written.
10. Click the **CFX-Solver** tab.
11. Select **File > Define Run**.
12. Under **Definition File**, click *Browse* .

13. Select `StaticMixer.def`, located in the working directory.

Obtaining a Solution Using ANSYS CFX-Solver Manager

ANSYS 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 ANSYS 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 ANSYS CFX-Solver Manager window (also adjustable).



One window shows the convergence history plots and the other displays text output from ANSYS 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`).

Start the Run

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

When ANSYS CFX-Solver Manager is launched automatically from ANSYS 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 ANSYS CFX-Solver without having to define settings and values.

Procedure

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

ANSYS 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 ANSYS CFX-Solver Manager operates.

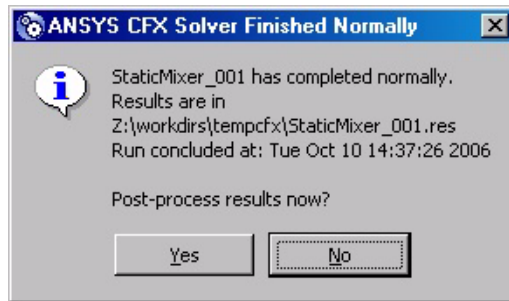
Note: Once the second iteration appears, data begins to plot. Plotting may take a long time depending on the amount of data to process. Let the process run.

Move from ANSYS CFX-Solver to ANSYS CFX-Post

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

Procedure

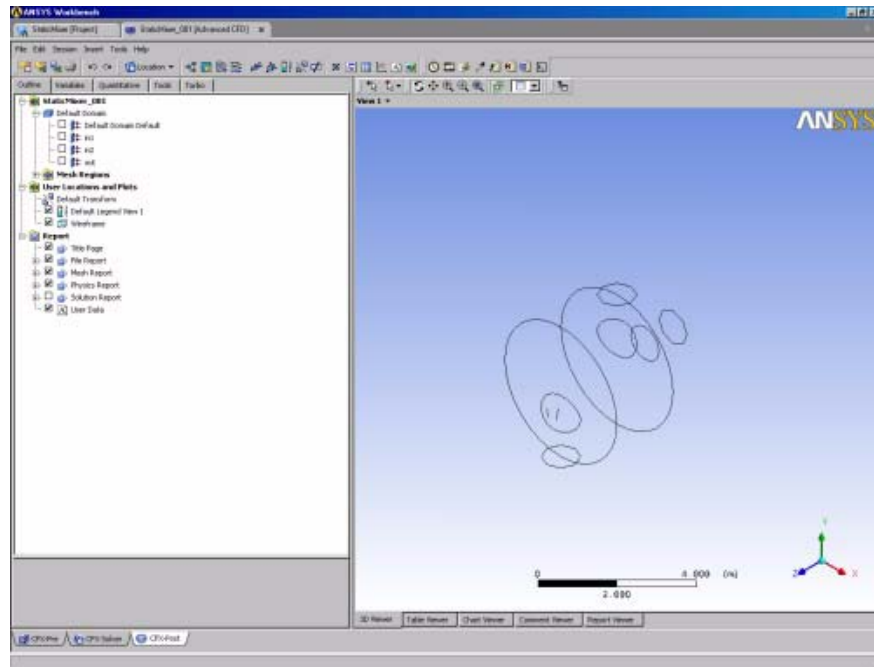
1. When ANSYS CFX-Solver is finished, click **Yes** to post-process the results.



After a short pause, ANSYS CFX-Post starts and ANSYS CFX-Solver Manager closes.

Viewing the Results in ANSYS CFX-Post

When ANSYS CFX-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 ANSYS CFX-Pre.

Workflow Overview

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

1. [Setting the Edge Angle for a Wireframe Object](#) (p. 44)
2. [Creating a Point for the Origin of the Streamline](#) (p. 45)
3. [Creating a Streamline Originating from a Point](#) (p. 46)
4. [Rearranging the Point](#) (p. 47)
5. [Configuring a Default Legend](#) (p. 47)
6. [Creating a Slice Plane](#) (p. 48)
7. [Defining Slice Plane Geometry](#) (p. 49)
8. [Configuring Slice Plane Views](#) (p. 49)
9. [Rendering Slice Planes](#) (p. 50)
10. [Coloring the Slice Plane](#) (p. 51)
11. [Moving the Slice Plane](#) (p. 51)
12. [Adding Contours](#) (p. 52)
13. [Working with Animations](#) (p. 53)

Setting the Edge Angle for a Wireframe Object

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

By default, ANSYS CFX-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** control the visibility of each object. Currently only the **Wireframe** and **Default Legend** objects have visibility selected.

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.

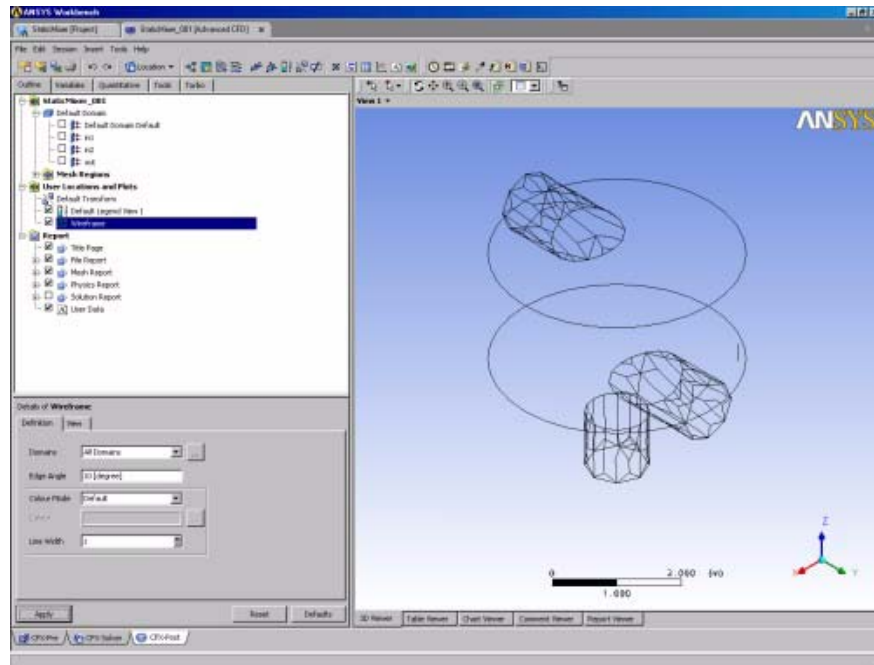
For this geometry, a setting of approximately 15° lets you view the model location without displaying an excessive amount of the surface mesh.

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

Procedure

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 on 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.
10. Right-click on a blank area anywhere in the viewer, select **Predefined Camera** from the shortcut menu and select **View Towards -X**.

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.

Procedure

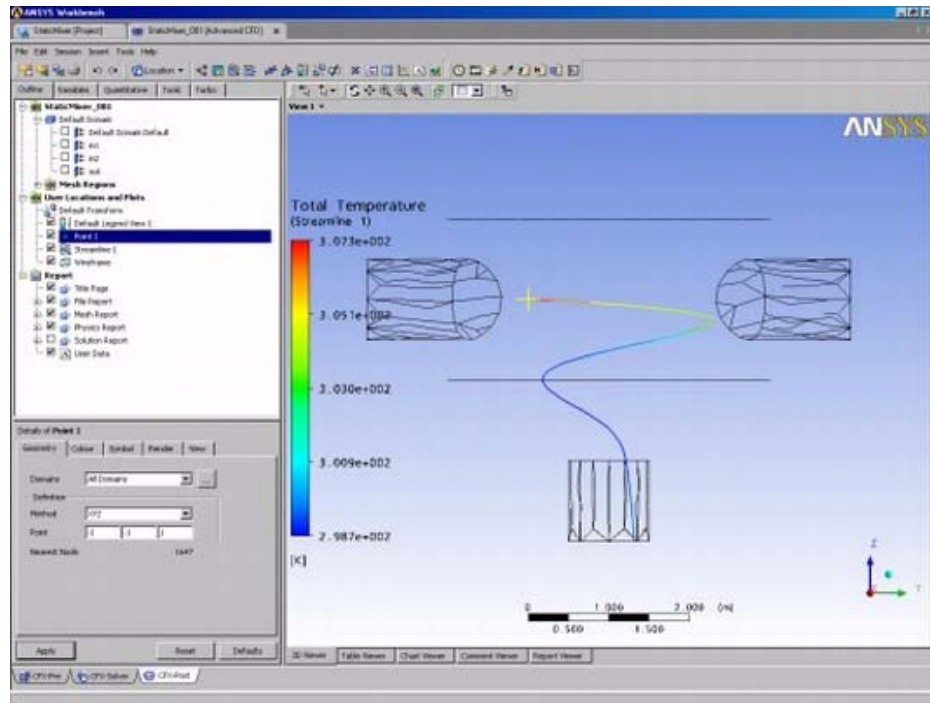
1. Select **Insert > Location > Point** from the main menu.
You can also use the toolbars to create a variety of objects. Later modules and tutorials explore this further.
2. Click **OK**.
This accepts the default name.
3. Under **Definition**, ensure that **Method** is set to **XYZ**.
4. Under **Point**, 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.

Creating a Streamline Originating from a Point

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

Procedure

- From the main menu, select **Insert > Streamline**.
You can also use the toolbars to create a variety of objects. Later modules and tutorials will explore this further.
- Click **OK**.
This accepts the default name.
- Under **Definition**, in **Start From**, ensure that `Point 1` is set.
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.
- Click the **Color** tab.
- Set **Mode** to `Variable`.
- Set **Variable** to `Total Temperature`.
- Set **Range** to `Local`.
- 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.



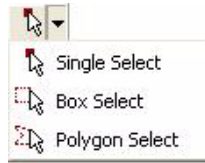
Rearranging the Point

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 Towards -X**) and by using **Zoom Box**  if you prefer to change the display.

Procedure

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 selection tools, click **Single Select**.



While in this mode, the normal behavior of the left mouse button is disabled.


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 Towards -X**.

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 plot¹.

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.

The default legend displays values for the last eligible plot that was opened in the details view. To maintain a legend definition during an ANSYS CFX-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**.

Procedure

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.

Creating a Slice Plane

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

In ANSYS CFX-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.

Procedure

1. From the main menu, select **Insert > Location > Plane** or click **Location > Plane**.
2. In the **New Plane** window, type: `Slice`
3. Click **OK**.
The **Geometry**, **Color**, **Render**, and **View** tabs let you switch between settings.
4. Click the **Geometry** tab.

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).

Procedure

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. Click **Apply**.
`Slice` displays under **User Locations and Plots**. Rotate the view to see the plane.

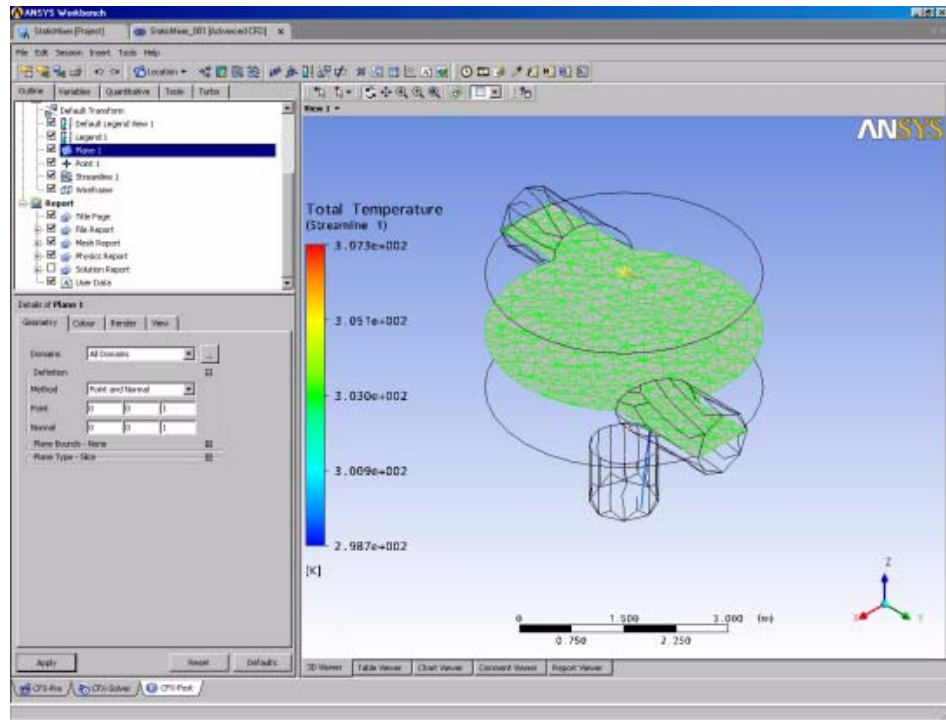
Configuring Slice Plane Views



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

Procedure

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.

Rendering Slice Planes

Render settings determine how the plane is drawn.

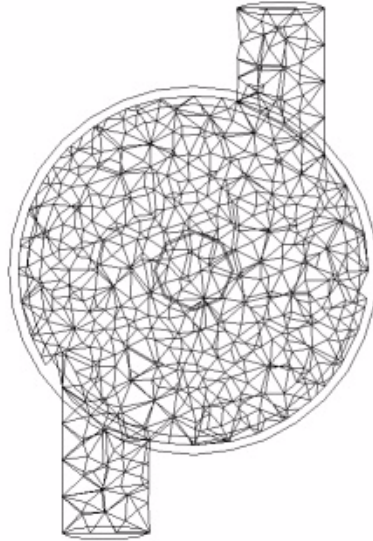
Procedure

1. In the **Details** pane for **Slice**, select the **Render** tab.
2. Clear **Draw Faces**.
3. Select **Draw Lines**.
4. Under **Draw 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 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 Towards -Z**.

The image shown below can be used for comparison with tutorial 2 (in the section [Creating a Slice Plane](#) (p. 68)), where a refined mesh is used.



Coloring the Slice Plane

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

Procedure

1. Apply the following settings to `slice`



Tab	Setting	Value
Color	Mode	Variable*
	Variable	Temperature
Render	Draw Faces	(Selected)
	Draw Lines	(Cleared)

*. 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.


Moving the Slice Plane

The plane can be moved to different locations.

- Procedure**
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 for *slice* by clearing the check box next to *slice* in the **Outline**.

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).

- Procedure**
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 **New Contour** dialog box is displayed.

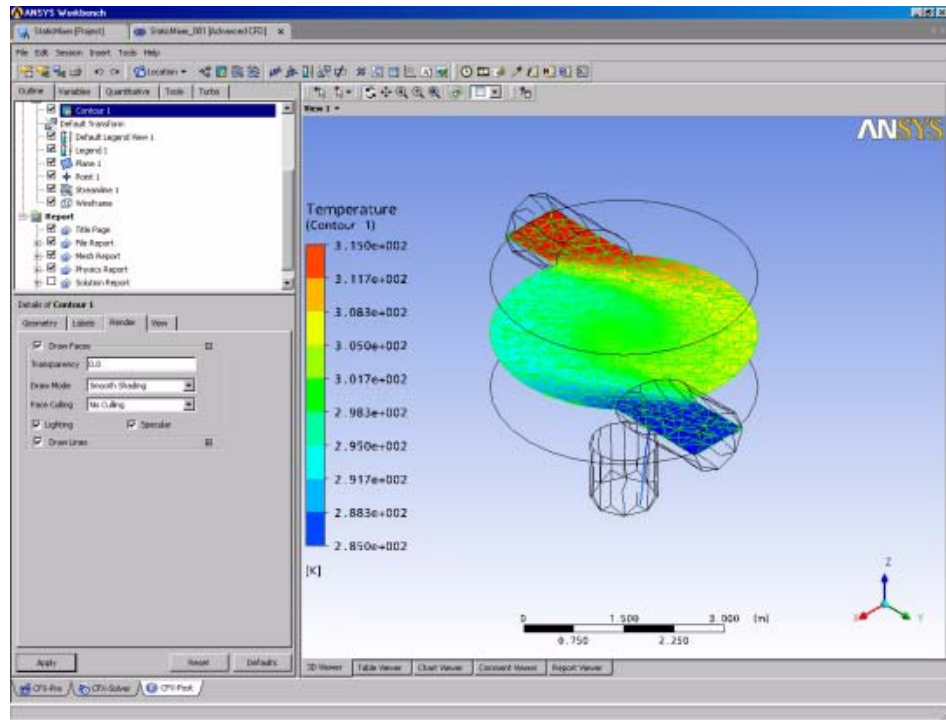
 3. Set **Name** to *slice Contour*.
 4. Click **OK**.
 5. Apply the following settings

Tab	Setting	Value
Geometry	Locations	slice
	Variable	Temperature
Render	Draw Faces	(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, on the **Render** tab under **Draw Faces** clear **Lighting** and click **Apply**.

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



Working with Animations

Animations build transitions between views for development of video files.

Workflow Overview

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

1. [Showing the Animation Dialog Box](#) (p. 53)
2. [Creating the First Keyframe](#) (p. 53)
3. [Creating the Second Keyframe](#) (p. 54)
4. [Viewing the Animation](#) (p. 55)
5. [Modifying the Animation](#) (p. 56)
6. [Saving to MPEG](#) (p. 57)

Showing the Animation Dialog Box

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

Procedure



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

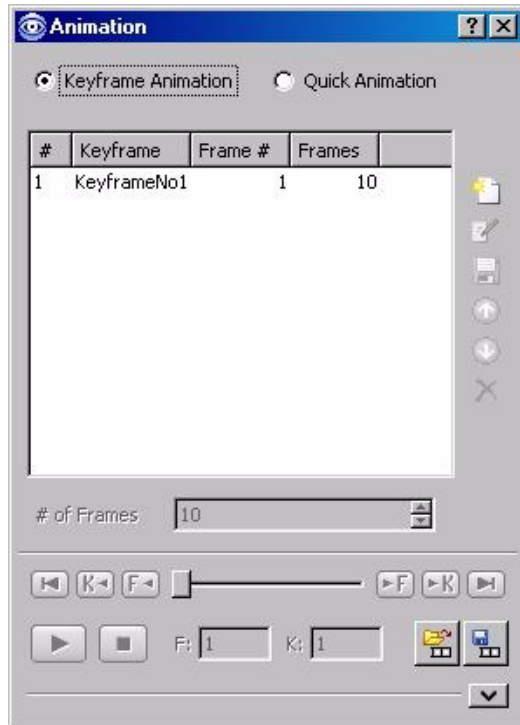
The **Animation** dialog box can be repositioned as required.

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.

Procedure


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**, clear the visibility of **Slice Contour** and select the visibility of **Slice**.
3. Select **Tools > Animation** or click **Animation** .
The **Animation** dialog box can be repositioned as required.
4. In the **Animation** dialog box, click **New** .
A new keyframe named `KeyframeNo1` is created. This represents the current image displayed in the viewer.



Creating the Second Keyframe

Keyframes are required in order to produce an animation.

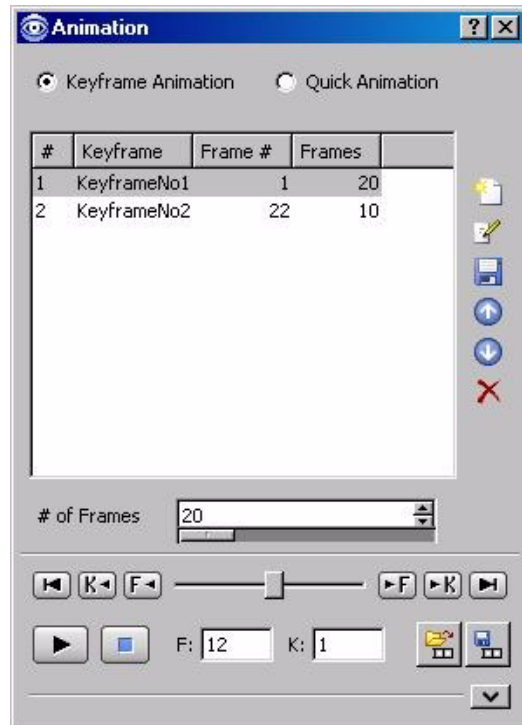
Procedure

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*.




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 keyframes listed beside the last keyframe is ignored.

Procedure

1. 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.

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.



Procedure

1. Apply the following settings to `Slice`


Tab	Setting	Value
Geometry	Point	0, 0, 1.99
Color	Mode	Variable
	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
	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`.

Saving to MPEG

By defining the geometry and then saving to MPEG, the results can be saved to a video file.

Procedure

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. Click **Save MPEG**.
3. Click *Browse*  next to **Save MPEG**.
4. Under **File name** type: `StaticMixer.mpg`
5. If required, set the path location to a different folder.
6. Click **Save**.
The MPEG file name (including path) is set. At this point, the animation has not yet been produced.
7. Click *Previous Keyframe* .
Wait a moment as the display updates the keyframe display.
8. Click *Play the animation* .
9. If prompted to overwrite an existing movie click **Overwrite**.
The animation plays and builds an MPEG file.
10. 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.
11. Click **Cancel** to close the dialog box.
12. Close the **Animation** dialog box.
13. Review the animation in third-party software as required.

Exiting ANSYS CFX-Post

When finished with ANSYS CFX-Post, exit the current window:

1. Select **File > Close** to close the current file.
2. If prompted to save, click **Close**.
3. Return to the Project page. Select **File > Close Project**.
4. Select **No**, then close Workbench.

Tutorial 2: Flow in a Static Mixer (Refined Mesh)

Introduction

This tutorial includes:

- [Tutorial 2 Features](#) (p. 60)
- [Overview of the Problem to Solve](#) (p. 60)
- [Defining a Simulation using General Mode in ANSYS CFX-Pre](#) (p. 61)
- [Obtaining a Solution Using Interpolation with ANSYS CFX-Solver](#) (p. 66)
- [Viewing the Results in ANSYS CFX-Post](#) (p. 68)

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

- [Setting the Working Directory](#) (p. 1)
- [Changing the Display Colors](#) (p. 2)

Unless you plan on running a session file, you should copy the sample files used in this tutorial from the installation folder for your software (<CFXROOT>/examples/) to your working directory. This prevents you from overwriting source files provided with your installation. If you plan to use a session file, please refer to [Playing a Session File](#) (p. 61).

Sample files used by this tutorial are:

- **StaticMixerRefMesh.gtm**
- **StaticMixerRef.pre**
- **StaticMixer.def**
- **StaticMixer_001.res**

Tutorial 2 Features

This tutorial addresses the following features of ANSYS CFX.

Component	Feature	Details	
ANSYS CFX-Pre	User Mode	General Mode	
	Simulation 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	
ANSYS CFX-Post	Plots	Planevolume	
		Slice Plane	
		Spherevolume	
	Other	Viewing the Mesh	

In this tutorial you will learn about:

- Using the General Mode of ANSYS CFX-Pre (this mode is used for more complex cases).
- Rerunning a problem with a refined mesh.
- Importing CCL 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 a Plane Volume locator and the Mesh Calculator to analyze mesh quality.

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 [Tutorial 1: Simulating Flow in a Static Mixer Using CFX in Standalone Mode](#) (p. 3). 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 ANSYS CFX-Pre and import the refined mesh. This tutorial introduces General Mode—the mode used for most tutorials—in ANSYS CFX-Pre. The physics for this tutorial are the same as for [Tutorial 1: Simulating Flow in a Static Mixer Using CFX in Standalone Mode](#) (p. 3); therefore, you can import the physics settings used in that tutorial to save time.

Defining a Simulation using General Mode in ANSYS CFX-Pre

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

Playing a Session File

If you want to skip past these instructions, and have ANSYS CFX-Pre set up the simulation automatically, you can select **Session > Play Tutorial** from the menu in ANSYS CFX-Pre, then run the appropriate session file. For details, see [Playing the Session File and Starting ANSYS CFX-Solver Manager](#) (p. 65). After you have played the session file, proceed to [Obtaining a Solution Using Interpolation with ANSYS CFX-Solver](#) (p. 66).

Workflow Overview

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

1. [Creating a New Simulation](#) (p. 61)
2. [Importing a Mesh](#) (p. 62)
3. [Importing CCL](#) (p. 62)
4. [Viewing Domain Settings](#) (p. 63)
5. [Viewing the Boundary Condition Setting](#) (p. 64)
6. [Defining Solver Parameters](#) (p. 64)
7. [Writing the Solver \(.def\) File](#) (p. 64)

As an alternative to these steps, you can also review [Playing the Session File and Starting ANSYS CFX-Solver Manager](#) (p. 65)

To begin this tutorial and create a new simulation in ANSYS CFX-Pre, continue from [Creating a New Simulation](#) (p. 61).

Creating a New Simulation

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

Note: Two procedures are documented. Depending on your installation of ANSYS CFX, follow either the Standalone procedure or the Workbench procedure.


Procedure in Standalone

1. If required, launch ANSYS CFX-Pre.
2. Select **File > New Simulation**.
3. Select **General** in the **New Simulation File** dialog box and click **OK**.
4. Select **File > Save Simulation As**.
5. Under **File name**, type `StaticMixerRef` and click **Save**.
6. Proceed to [Importing a Mesh](#) (p. 62).

Procedure in Workbench

1. If required, launch ANSYS Workbench.
2. Click **Empty Project**.

The Project page appears displaying an unsaved project.

3. Select **File > Save** or click *Save*  .
4. If required, set the path location to your working folder.
5. Under **File name**, type `StaticMixerRef` and click **Save**.
6. Click **Start CFX-Pre** under **Advanced CFD** on the left hand task bar.
7. Select **File > New Simulation**.
8. Click **General** in the **New Simulation File** window, and then click **OK**.
9. Select **File > Save Simulation As**.
10. Under **File name**, type `StaticMixerRef` and click **Save**.

Importing a Mesh

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

An assembly is a group of mesh regions that are topologically connected. Each assembly can contain only one mesh, but multiple assemblies are permitted. The `Mesh` tree shows the regions in `Assembly` in a tree structure. The level below `Assembly` displays 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.

Procedure

1. Select **File > Import Mesh** or right-click `Mesh` and select **Import Mesh**.
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**.
4. Right-click a blank area in the viewer and select **Predefined Camera > Isometric View (Z up)** from the shortcut menu.

Importing CCL

Since the physics for this simulation is very similar to that for Tutorial 1, you can save time by importing the settings used there.

The CCL 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.

The physics for a simulation can be saved to a CCL (CFX Command Language) file at any time by selecting **File > Export CCL**. However, a number of other files can also be used as sources to import CCL including:

- Simulation files (`*.cfx`)
- Results files (`*.res`)
- Definition files (`*.def`)

Note: If you import CCL that references non-existent mesh regions, you will get errors.

Procedure

1. Select **File > Import CCL**.
The **Import CCL** dialog box appears.
2. Under **Import Method**, select **Append**.
Replace is useful if you have defined physics and want to update or replace them with newly imported physics.
3. Under **File type**, select `CFX-Solver 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.

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

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`.

Viewing Domain Settings

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

Various domain settings can be set. These include:

- **General Options**
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.

Procedure

1. On the **Outline** tree view, under `Simulation`, double-click `Default Domain`.
The domain `Default Domain` is opened for editing.
2. Click **General Options** 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**.

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.

Procedure

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**.

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 ANSYS 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.

Procedure

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

Setting	Value
Advection Scheme > Option	High Resolution
Convergence Control > Max. Iterations*	150
Convergence Control > Fluid Timescale Control > Timescale Control	Physical Timescale
Convergence Control > Fluid Timescale Control > Physical Timescale	0.5 [s]

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

3. Click **Apply**.
4. Click the **Advanced Options** tab.
5. Ensure that **Global Dynamic Model Control** is selected.
6. Click **OK**.

Writing the Solver (.def) File

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

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

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

Procedure

1. Click *Write Solver File* .

The **Write Solver 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
Quit CFX-Pre*	(Selected)

*. If using ANSYS CFX-Pre in Standalone Mode.


4. Ensure **Start Solver Manager** is selected and click **Save**.
5. If you are notified that the file already exists, click **Overwrite**.
6. If prompted, click **Yes** or **Save & Quit** to save **StaticMixerRef.cfx**.
The definition file (**StaticMixerRef.def**) and the simulation file (**StaticMixerRef.cfx**) are created. ANSYS CFX-Solver Manager automatically starts and the definition file is set in the **Definition File** box of **Define Run**.
7. Proceed to [Obtaining a Solution Using Interpolation with ANSYS CFX-Solver](#) (p. 66).

Playing the Session File and Starting ANSYS CFX-Solver Manager

If you have performed all the tasks in the previous steps, proceed directly to [Obtaining a Solution Using Interpolation with ANSYS CFX-Solver](#) (p. 66).



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

Procedure in Standalone

1. If required, launch ANSYS CFX-Pre.
2. Select **Session > Play Tutorial**.
3. Select **StaticMixerRef.pre**.
4. Click **Open**.
A definition file is written.
5. Select **File > Quit**.
6. Launch ANSYS CFX-Solver Manager from CFX Launcher.
7. After ANSYS CFX-Solver starts, select **File > Define Run**.
8. Under **Definition File**, click *Browse* .
9. Select **StaticMixerRef.def**, located in the working directory.

10. Proceed to [Obtaining a Solution Using Interpolation with ANSYS CFX-Solver](#) (p. 66).

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

Obtaining a Solution Using Interpolation with ANSYS CFX-Solver

Two windows are displayed when ANSYS 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 ANSYS CFX-Solver Manager window (also adjustable).

Workflow Overview

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

1. [Interpolating the Results and Starting the Run](#) (p. 66)
2. [Confirming Results](#) (p. 67)
3. [Moving from ANSYS CFX-Solver to ANSYS CFX-Post](#) (p. 67)

Interpolating the Results and Starting the Run


In the ANSYS CFX-Solver Manager, **Define Run** is visible and **Definition File** has automatically been set to the definition file from ANSYS CFX-Pre: `StaticMixerRef.def`. You want to make use of the results from Tutorial 1, but the two meshes are not identical. The initial values file needs to have its data interpolated onto the new mesh associated with the definition file.

The ANSYS CFX-Solver supports automatic interpolation that will be used in the following steps:

Tutorial 2: Flow in a Static Mixer (Refined Mesh): Obtaining a Solution Using Interpolation with ANSYS CFX-Solver

The values from `StaticMixer_001.res` will be interpolated onto the definition file's mesh when the run is started. The results from `StaticMixer_001.res` will be used as the initial guess for this simulation (rather than Solver defaults) because you have set the initialization for all variables in ANSYS CFX-Pre to **Automatic** or **Automatic with Value**.

Procedure

1. Under **Initial Values File**, click *Browse* .
2. Select the results file from Tutorial 1: `StaticMixer_001.res`
If you did not complete the first tutorial, you can use `StaticMixer_001.res` from your working directory.
3. Click **Open**.
4. Select **Interpolate Initial Values onto Def File Mesh**.
5. Click **Start Run**.

Note: The message **Finished interpolation successfully** appears relatively quickly. Convergence information is plotted once the second outer loop iteration is complete.

Confirming Results

When interpolation is successful, specific information appears in the text screen of ANSYS CFX-Solver.

To confirm that the interpolation was successful, look in the text pane in ANSYS 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:

```
CFD Solver finished: Tue Oct 19 08:06:45 2004  
CFD Solver wall clock seconds: 1.7100E+02  
Execution terminating:  
all residual  
are below their target criteria
```

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


Procedure

1. When the run finishes and asks if you want to post-process the results, click **No** to keep ANSYS CFX-Solver open. Review the results on the **Out File** tab for details on the run results.

Moving from ANSYS CFX-Solver to ANSYS CFX-Post

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

Procedure

1. Select **Tools > Post-Process Results** or click *Post-Process Results*  in the toolbar.
2. If using ANSYS CFX-Solver in Standalone Mode, select **Shut down Solver Manager**. This forces Standalone ANSYS CFX-Solver to close when finished. This option is not required in Workbench.
3. Click **OK**.

After a short pause, ANSYS CFX-Post starts.

Viewing the Results in ANSYS CFX-Post

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

Creating a Slice Plane

More information exists for use by ANSYS CFX-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.

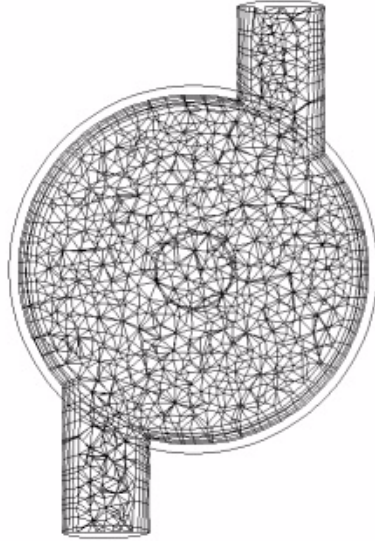
Procedure

1. Right-click a blank area in the viewer and select **Predefined Camera > Isometric View (Z up)**.
2. From the main menu, 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]
Render	Draw Faces	(Cleared)
	Draw Lines	(Selected)

5. Click **Apply**.
6. Right-click a blank area in the viewer and select **Predefined Camera > View Towards -Z**.
7. Click **Zoom Box** .
8. Zoom in on the geometry to view it in greater detail.

Compare the on-screen image with the equivalent picture from tutorial 1 (in the section [Rendering Slice Planes](#) (p. 22)).



Coloring the Slice Plane

Here, you will color the plane by temperature.

Procedure

1. Apply the following settings

Tab	Setting	Value
Color	Mode*	Variable
	Variable	Temperature
	Range	Global
Render	Draw Faces	(Selected)
	Draw Lines	(Cleared)


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

2. Click **Apply**.

Loading Results from Tutorial 1 for Comparison

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

Procedure

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 two frames. Under **Results file option**, select **Add to current results**.
4. Select the **Offset in Y direction** check box.
5. Under **Additional actions**, ensure that the **Clear user state before loading** check box is cleared.
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`.
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. 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.
11. Set **Edge Angle** to 30 [degree].
12. Click **Apply**.

Creating a Second Slice Plane

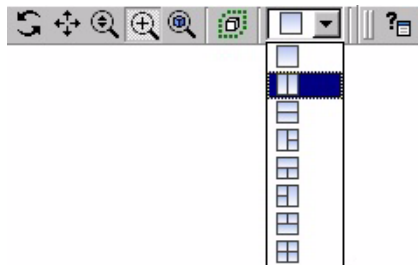
Procedure

1. In the tree view, right-click the plane named `Slice` and select **Duplicate**.
2. Click **OK** to accept the default name `Slice 1`.
3. In the tree view, double-click the plane named `Slice 1`.
4. On the **Geometry** tab, set **Domains** to `Default Domain 1`.
5. On the **Color** tab, ensure that **Range** is set to `Global`.
6. Click **Apply**.
7. Double-click `Slice` and make sure that **Range** is set to `Global`.


Comparing Slice Planes using Multiple Views

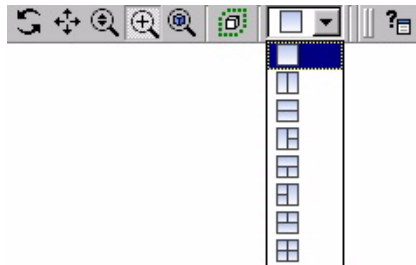
Procedure

1. Select the option with the two vertical rectangles. Notice that the Viewer now has two separate views.



The visibility status of each object is maintained separately for each view or figure that can be displayed in a given viewport. This allows some planes to be shown while others are hidden.

2. Click in the viewport that is set to show **View 1**, then clear the visibility check box for `slice` in the **Outline** tree view and ensure that the visibility check box for `slice 1` is selected.
3. Click in the viewport that is set to show **View 2**, then select the visibility check box for `slice` and ensure that the visibility check box for `slice 1` is cleared.
4. In the tree view, double-click `StaticMixer_001` and clear **Apply Translation**.
5. Click **Apply**.
6. In the viewer toolbar, click *Synchronise Active Views* . Notice that both views move in the same way and are zoomed in at the same level.
7. Right-click in the viewer and select **Predefined Camera > View Towards -Z**. Note the difference in temperature distribution.
8. To return to a single viewport, select the option with a single rectangle.



9. Right-click `slice 1` in the tree view and select **Delete**.
10. Ensure that the visibility check box for `slice` is cleared.
11. Right-click `StaticMixer_001` in the tree view and select **Unload**.

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.

Procedure

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. Since the boundary location geometry was defined in ANSYS 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	Draw Faces	(Cleared)
	Draw Lines	(Selected)
	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.

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.

Procedure

1. From the main menu, 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*	0.08, 0, -2
	Definition > Radius	0.14 [m]
	Definition > Mode	Below Intersection
	Inclusive†	(Cleared)
Color	Color	Red
Render	Draw Faces > Transparency	0.3
	Draw Lines	(Selected)
	Draw Lines > Line Width	1
	Draw Lines > Color Mode	User Specified
	Draw Lines > Line Color	Grey

*. The z slider's minimum value corresponds to the minimum z value of the entire geometry, which, in this case, occurs at the outlet.

†. 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

- Click **Apply**.

Viewing the Surface Mesh on the Mixer Body

Procedure

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

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

- Click **Apply**.

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.

Procedure

- From the main menu, select **Insert > Location > Plane** or under **Location**, click **Plane**.
- In the **Insert Plane** dialog box, type `slice 2` and click **OK**.
- Apply the following settings

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

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

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 ANSYS CFX-Post and check the mesh quality before running the `.def` file in the solver.

Procedure

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.


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°.

Procedure

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 main menu, 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*
	Definition > Mode	Above Value
	Definition > Value	140 [degree]
	Inclusive†	(Selected)

*. Select `Maximum Face Angle` from the larger list of variables available by clicking  to the right of the **Variable** box.

†. 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.

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.

Procedure

1. From the main menu, 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**.

Quitting ANSYS CFX-Post

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

Procedure in Standalone

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

Procedure in Workbench

1. When you are finished, select **File > Close** to close the current file.
2. Click **Close** if prompted to save.
3. Return to the Project page. Select **File > Close Project**.
4. Select **No**, then close Workbench.

Tutorial 3: Flow in a Process Injection Mixing Pipe

Introduction

This tutorial includes:

- [Tutorial 3 Features](#) (p. 78)
- [Overview of the Problem to Solve](#) (p. 78)
- [Defining a Simulation using General Mode in ANSYS CFX-Pre](#) (p. 79)
- [Obtaining a Solution Using ANSYS CFX-Solver Manager](#) (p. 87)
- [Viewing the Results in ANSYS CFX-Post](#) (p. 88)

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

- [Setting the Working Directory](#) (p. 1)
- [Changing the Display Colors](#) (p. 2)

Unless you plan on running a session file, you should copy the sample files used in this tutorial from the installation folder for your software (<CFXROOT>/examples/) to your working directory. This prevents you from overwriting source files provided with your installation. If you plan to use a session file, please refer to [Playing a Session File](#) (p. 79).

Sample files referenced by this tutorial include:

- **InjectMixer.pre**
- **InjectMixer_velocity_profile.csv**
- **InjectMixerMesh.gtm**

Tutorial 3 Features

This tutorial addresses the following features of ANSYS CFX.

Component	Feature	Details	
ANSYS CFX-Pre	User Mode	General Mode	
	Simulation 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	
ANSYS CFX-Post	Plots	default Locators	
		Outline Plot (Wireframe)	
		Slice Plane	
		Streamline	
	Other		Changing the Color Range
			Expression Details View
		Legend	
		Viewing the Mesh	

In this tutorial you will learn about:

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

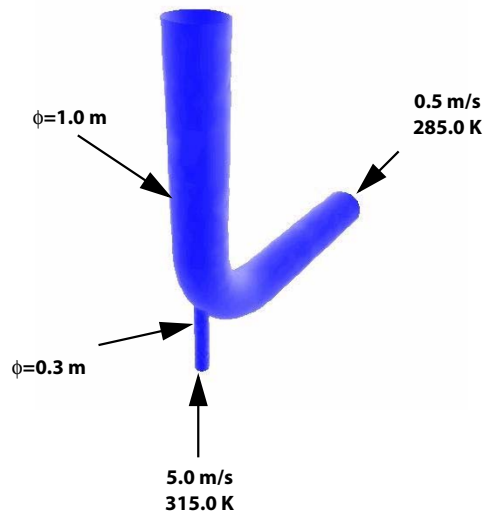
Overview of the Problem to Solve

In this tutorial, you establish a general workflow for analyzing the flow of fluid into and out of an injection pipe. This tutorial is important because it uses a specific problem to teach the general approach taken when working with an existing mesh.

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.

Figure 1 Injection Mixing Pipe



Defining a Simulation using General Mode in ANSYS CFX-Pre

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

Playing a Session File

If you wish to skip past these instructions, and have ANSYS CFX-Pre set up the simulation automatically, you can select **Session > Play Tutorial** from the menu in ANSYS CFX-Pre, then run the appropriate session file. For details, see [Playing the Session File and Starting ANSYS CFX-Solver Manager](#) (p. 87). After you have played the session file, proceed to [Obtaining a Solution Using ANSYS CFX-Solver Manager](#) (p. 87).

Workflow Overview

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

1. [Creating a New Simulation](#) (p. 80)
2. [Importing a Mesh](#) (p. 80)
3. [Setting Temperature-Dependent Material Properties](#) (p. 81)
4. [Plotting an Expression](#) (p. 82)

5. [Evaluating an Expression](#) (p. 82)
6. [Modify Material Properties](#) (p. 82)
7. [Creating the Domain](#) (p. 82)
8. [Creating the Side Inlet Boundary Conditions](#) (p. 83)
9. [Creating the Main Inlet Boundary Conditions](#) (p. 84)
10. [Creating the Main Outlet Boundary Condition](#) (p. 85)
11. [Setting Initial Values](#) (p. 85)
12. [Setting Solver Control](#) (p. 85)
13. [Writing the Solver \(.def\) File](#) (p. 86)

Creating a New Simulation


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

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

Procedure in Standalone

1. If required, launch ANSYS CFX-Pre.
2. Select **File > New Simulation**.
3. Ensure **General** is selected and click **OK**.
4. Select **File > Save Simulation As**.
5. Under **File name**, type `InjectMixer`.
6. Click **Save**.
7. Proceed to [Importing a Mesh](#) (p. 80).

Procedure in Workbench

1. If required, launch ANSYS Workbench.
2. Click **Empty Project**.
The Project page will appear displaying an unsaved project.
3. Select **File > Save** or click *Save*  .
4. If required, set the path location to your working folder.
5. Under **File name**, type `InjectMixer`.
6. Click **Save**.
7. Click **Start CFX-Pre** under **Advanced CFD** on the left hand task bar.
8. Select **File > New Simulation**.
9. Click **General** in the **New Simulation File** window and then click **OK**.
10. Select **File > Save Simulation As**.
11. Under **File name**, type `InjectMixer`.
12. Click **Save**.

Importing a Mesh

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

- Procedure**
1. Select **File > Import Mesh**.
 2. From your tutorial directory, select **InjectMixerMesh.gtm**.
 3. Click **Open**.
 4. Right-click a blank area in the viewer and select **Predefined Camera > Isometric View (Y up)** from the shortcut menu.

Setting Temperature-Dependent Material Properties

You will create an expression for viscosity as a function of temperature and then use this expression to modify the properties of the library material: **Water**.

Viscosity will be made to vary linearly with temperature between the following conditions:

- $\mu = 1.8E-03 \text{ N s m}^{-2}$ at $T=275.0 \text{ K}$
- $\mu = 5.45E-04 \text{ N s m}^{-2}$ at $T=325.0 \text{ K}$

The variable **T** (Temperature) is a ANSYS CFX System Variable recognized by ANSYS 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 ANSYS CFX-Pre.

- Procedure**
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 **New**.
 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)

Plotting an Expression

Procedure

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.

Evaluating an Expression


Procedure

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**.
The value of `visT` for the given value of `T` appears in the **Value** field.

Modify Material Properties

Default material properties (such as those of `water`) can be modified when required.

Procedure


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**.

Creating the Domain

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

Both **General Options** 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.

Procedure

1. Select **Insert > Domain** from the main menu or click *Domain* .
2. In the **Insert Domain** dialog box, type `InjectMixer`.
3. Click **OK**.
4. Apply the following settings

Tab	Setting	Value
General Options	Basic Settings > Location	B1.P3
	Basic Settings > Fluids List	Water
	Domain Models > Pressure > Reference Pressure	0 [atm]

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


Setting	Value
Heat Transfer > Option	Thermal Energy

7. Click **OK**.

Creating the Side Inlet Boundary Conditions

The side inlet boundary condition needs to be defined.

Procedure

1. Select **Insert > Boundary Condition** from the main menu or click *Boundary Condition* .
2. Set **Name** to `side inlet`.
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
	Normal Speed	5 [m s ⁻¹]
	Heat Transfer > Option	Static Temperature
	Static Temperature	315 [K]



5. Click **OK**.

Creating the Main Inlet Boundary Conditions

The main inlet boundary condition needs to be defined. This inlet is defined using a velocity profile found in the example directory. Profile data needs to be initialized before the boundary condition 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.

Procedure

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 Condition** from the main menu or click *Boundary Condition* .
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 condition 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
	Static Temperature	285 [K]
Plot Options	Boundary Contour	(Selected)
	Profile Variable	W


12. Click **OK**.

- Zoom into the main inlet to view the inlet velocity contour.

Creating the Main Outlet Boundary Condition

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

Procedure


- Select `Insert > Boundary Condition` from the main menu or click *Boundary Condition* .
- Set **Name** to `outlet`.
- Click **OK**.
- 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]

- Click **OK**.


Setting Initial Values

Procedure

- Click *Global Initialization* .
- Select **Turbulence Eddy Dissipation**.
- Click **OK**.

Setting Solver Control

Procedure

- Click *Solver Control* .
- Apply the following settings

Tab	Setting	Value
Basic Settings	Advection Scheme > Option	Specified Blend Factor
	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]
	Convergence Criteria > Residual Type	RMS
	Convergence Criteria > Residual Target	1.E-4*

*. 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**.

Writing the Solver (.def) File

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

Procedure

1. Click *Write Solver File* .
The **Write Solver File** dialog box appears.
2. Apply the following settings:

Setting	Value
File name	InjectMixer.def
Quit CFX-Pre*	(Selected)

*. If using ANSYS CFX-Pre in Standalone Mode.


3. Ensure **Start Solver Manager** is selected and click **Save**.
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 folder if you have copied it there.
5. If prompted, click **Yes** or **Save & Quit** to save **InjectMixer.cfx**.
The definition file (**InjectMixer.def**), mesh file (**InjectMixer.gtm**) and the simulation file (**InjectMixer.cfx**) are created. ANSYS CFX-Solver Manager automatically starts and the definition file is set in the **Definition File** box of **Define Run**.
6. Proceed to [Obtaining a Solution Using ANSYS CFX-Solver Manager](#) (p. 87).

Playing the Session File and Starting ANSYS CFX-Solver Manager



If you have performed all the tasks in the previous steps, proceed directly to [Obtaining a Solution Using ANSYS CFX-Solver Manager](#) (p. 87).

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

Procedure in Standalone

1. If required, launch ANSYS CFX-Pre.
2. Select **Session > Play Tutorial**.
3. Select **InjectMixer.pre**.
4. Click **Open**.
A definition file is written.
5. Select **File > Quit**.
6. Launch ANSYS CFX-Solver Manager from CFX Launcher.
7. After ANSYS CFX-Solver starts, select **File > Define Run**.
8. Under **Definition File**, click *Browse*  .
9. Select **InjectMixer.def**, located in the working directory.
10. Proceed to [Obtaining a Solution Using ANSYS CFX-Solver Manager](#) (p. 87).

Procedure in ANSYS Workbench


1. If required, launch ANSYS Workbench.
2. Click **Empty Project**.
3. Select **File > Save** or click *Save*  .
4. Under **Filename**, type `InjectMixer`.
5. Click **Save**.
6. Click **Start CFX-Pre**.
7. Select **Session > Play Tutorial**.
8. Select **InjectMixer.pre**.
9. Click **Open**.
A definition file is written.
10. Click the **CFX-Solver** tab.
11. Select **File > Define Run**.
12. Under **Definition File**, click *Browse*  .
13. Select **InjectMixer.def**, located in the working directory.

Obtaining a Solution Using ANSYS CFX-Solver Manager

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

1. Click **Start Run**.
2. Click **No** to close the message box that appears when the run ends.

Moving from ANSYS CFX-Solver Manager to ANSYS CFX-Post

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

Viewing the Results in ANSYS CFX-Post

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

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. 88)
2. [Creating and Modifying Streamlines](#) (p. 88)
3. [Modifying Streamline Color Ranges](#) (p. 89)
4. [Coloring Streamlines with a Constant Color](#) (p. 89)
5. [Duplicating and Modifying a Streamline Object](#) (p. 90)
6. [Examining Turbulent Kinetic Energy](#) (p. 90)

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.


Procedure

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

Creating and Modifying Streamlines

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.

Procedure

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
	Definition > Start From	main inlet

- Click **Apply**.
- 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.

Modifying Streamline Color Ranges

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

Procedure

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

Tab	Setting	Value
Color	Range	Local

- 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.
- 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.

- Click **Apply**.
The streamlines are colored using the specified range of velocity values.

Coloring Streamlines with a Constant Color

- 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**.

Duplicating and Modifying a Streamline Object

Any object can be duplicated to create a copy for modification without altering the original.

Procedure

1. Right-click `MainStream` and select **Duplicate** from the shortcut menu.
2. In the **Name** window, type `SideStream`.
3. Click **OK**.
4. Double click on 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)**.

Examining Turbulent Kinetic Energy

A common way of viewing various quantities within the domain is to use a slice plane, as demonstrated in this module.

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


Procedure

1. Clear visibility for 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.
 - a. From the main menu, select **Insert > Location > Plane** and click **OK**.
 - b. In the **Details** view set **Definition > Method** to `YZ Plane` and **X** to `0 [m]`.
 - c. 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
	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.

Exiting ANSYS CFX-Post

When finished with ANSYS CFX-Post exit the current window.

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

Procedure in Standalone

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

Procedure in Workbench

1. When you are finished, select **File > Close** to close the current file.
2. Click **Close** if prompted to save.
3. Return to the Project page. Select **File > Close Project**.
4. Select **No**, then close Workbench.

Tutorial 4: Flow from a Circular Vent

Introduction

This tutorial includes:

- [Tutorial 4 Features](#) (p. 94)
- [Overview of the Problem to Solve](#) (p. 95)
- [Defining a Steady-State Simulation in ANSYS CFX-Pre](#) (p. 95)
- [Obtaining a Solution to the Steady-State Problem](#) (p. 99)
- [Defining a Transient Simulation in ANSYS CFX-Pre](#) (p. 100)
- [Obtaining a Solution to the Transient Problem](#) (p. 104)
- [Viewing the Results in ANSYS CFX-Post](#) (p. 105)

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

- [Setting the Working Directory](#) (p. 1)
- [Changing the Display Colors](#) (p. 2)

Unless you plan on running a session file, you should copy the sample files used in this tutorial from the installation folder for your software (<CFXROOT>/examples/) to your working directory. This prevents you from overwriting source files provided with your installation. If you plan to use a session file, please refer to [Playing a Session File](#) (p. 95).

Sample files referenced by this tutorial include:

- **CircVent.pre**
- **CircVentIni.pre**
- **CircVentIni_001.res**
- **CircVentMesh.gtm**
- **CircVentIni.cfx**
- **CircVentIni.gtm**

Tutorial 4 Features

This tutorial addresses the following features of ANSYS CFX.

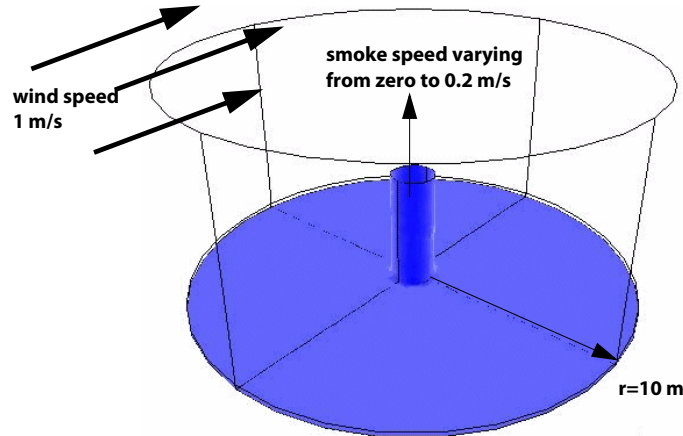
Component	Feature	Details
ANSYS CFX-Pre	User Mode	General Mode
	Simulation Type	Steady State
		Transient
	Fluid Type	General Fluid
	Domain Type	Single Domain
	Turbulence Model	k-Epsilon
	Boundary Conditions	Inlet (Subsonic)
		Opening
		Wall: No-Slip
	Timestep	Auto Time Scale
Transient Example		
	Transient Results File	
ANSYS CFX-Post	Plots	Animation
		Isosurface
	Other	Auto Annotation
		MPEG Generation
		Printing
		Time Step Selection
		Title/Text
Transient Animation		

In this tutorial you will learn about:

- Setting up a transient problem in ANSYS CFX-Pre.
- Using an opening type boundary condition in ANSYS CFX-Pre.
- Modeling smoke using additional variables in ANSYS CFX-Pre.
- Visualizing a smoke plume using an Isosurface in ANSYS CFX-Post.
- Creating an image for printing, and generating an MPEG file in ANSYS CFX-Post.

Overview of the Problem to Solve

In this example, a chimney stack releases smoke which is dispersed into the atmosphere with an oncoming side wind. Unlike previous tutorials, which were steady-state, this example is time-dependent. Initially, no smoke is being released. In the second part of the tutorial, the chimney starts to release smoke and it shows how the plume of smoke above the chimney develops with time.



Defining a Steady-State Simulation in ANSYS CFX-Pre

This section describes the step-by-step definition of the flow physics in ANSYS CFX-Pre for a steady-state simulation with no smoke being produced by the chimney. The results from this simulation will be used as the initial guess for the transient simulation.

Playing a Session File

If you wish to skip past these instructions, and have ANSYS CFX-Pre set up the simulation automatically, you can select **Session > Play Tutorial** from the menu in ANSYS CFX-Pre, then run the session file: `CircVentIni.pre`. After you have played the session file as described in earlier tutorials under [Playing the Session File and Starting ANSYS CFX-Solver Manager](#) (p. 87), proceed to [Obtaining a Solution to the Steady-State Problem](#) (p. 99).

Creating a New Simulation

1. Start ANSYS CFX-Pre.
2. Select **File > New Simulation**.
3. Select **General** and click **OK**.
4. Select **File > Save Simulation As**.
5. Under **File name**, type `CircVentIni`.
6. Click **Save**.


Importing the Mesh

1. Select **File > Import Mesh**.
2. From your working directory, select `CircVentMesh.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.

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 the user from having to set up timevalue interpolation in the transient simulation.

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

Creating the Domain

The fluid domain will be created that includes the additional variable.

To Create a New Domain


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

Tab	Setting	Value
General Options	Fluids List	Air at 25 C
	Reference Pressure	0 [atm]
Fluid Models	Heat Transfer > Option	None
	Additional Variable Details > smoke	(Selected)
	Additional Variable Details > smoke > Kinematic Diffusivity	(Selected)
	Additional Variable Details > smoke > Kinematic Diffusivity > Kinematic Diffusivity	1.0E-5 [m^2 s^-1]

3. Click **OK**.


Creating the Boundary Conditions

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 such 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.

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 condition and clear **Show Inlet Markers** on the **Boundary Marker Options** tab of **Labels and Markers** (accessible by clicking *Label and Marker Visibility* ).

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


Inlet Boundary

1. Select **Insert > Boundary Condition** from the main menu or click *Boundary Condition* .
2. Under **Name**, type `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 > Value	0.05
	Turbulence > Eddy Len. Scale	0.25 [m]
	Additional Variables > smoke > Option	Value
Additional Variables > smoke > Value	0 [kg m ⁻³]	

5. Click **OK**.


Opening Boundary

1. Select **Insert > Boundary Condition** from the main menu or click *Boundary Condition* .
2. Under **Name**, type *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 > Value	0.05
	Turbulence > Eddy Len. Scale	0.25 [m]
	Additional Variables > smoke > Option	Value
	Additional Variables > smoke > Value	0 [kg m ⁻³]

5. Click **OK**.


Inlet for the Vent

1. Select **Insert > Boundary Condition** from the main menu or click *Boundary Condition* .
2. Under **Name**, type *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
	Additional Variables > smoke > Option	Value
	Additional Variables > smoke > Value	0 [kg m ⁻³]


5. Click **OK**.

Setting Initial Values

1. Click *Global Initialization* .
2. Select **Turbulence Eddy Dissipation**.
3. Click **OK**.

Setting Solver Control


ANSYS CFX-Solver has the ability to calculate physical timestep 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. Click **OK**.

Writing the Solver (.def) File

1. Click *Write Solver File* .
2. Apply the following settings

Setting	Value
File name	CircVentIni.def
Quit CFX-Pre *	(Selected)

*. If using ANSYS CFX-Pre in Standalone Mode.

3. Ensure **Start Solver Manager** is selected and click **Save**.
4. Quit ANSYS CFX-Pre, saving the simulation (**.cfx**) file.

Obtaining a Solution to the Steady-State Problem

When ANSYS CFX-Pre has shut down and ANSYS CFX-Solver Manager has started, you can obtain a solution to the CFD problem by using the following procedure.

1. Click **Start Run**.
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 variable smoke is also plotted but registers no values since it is not initialized.
2. Click **No** to close the completion message, since you do not need to view the results in ANSYS CFX-Post.
3. If using Standalone Mode, quit ANSYS CFX-Solver Manager.
You will now reload the simulation into ANSYS CFX-Pre to define the transient simulation.

Defining a Transient Simulation in ANSYS CFX-Pre

In this part of the tutorial, you alter the simulation settings used for the steady-state calculation to set up the model for the transient calculation in ANSYS CFX-Pre.

Playing a Session File


If you wish to skip past these instructions, and have ANSYS CFX-Pre set up the simulation automatically, you can select **Session > Play Tutorial** from the menu in ANSYS CFX-Pre, then run the session file: `circVent.pre`. After you have played the session file as described in earlier tutorials under [Playing the Session File and Starting ANSYS CFX-Solver Manager](#) (p. 87), proceed to [Obtaining a Solution to the Transient Problem](#) (p. 104).

Opening the Existing Simulation


1. Start ANSYS CFX-Pre.
2. Select **File > Open Simulation**.
3. If required, set the path location to the tutorial folder.
4. Select the simulation file `circVentIni.cfx`.
5. Click **Open**.
6. Select **File > Save Simulation As**.
7. Change the name to `circVent.cfx`.
8. Click **Save**.

Modifying the Simulation Type

In this step you will make the problem transient. Later, you will set the concentration of smoke to rise exponentially with time, so it is necessary to ensure that the interval between the timesteps is smaller at the beginning of the simulation than at the end.

1. Click *Simulation Type* .
2. Apply the following settings

Tab	Setting	Value
Basic Settings	Simulation Type > Option	Transient
	Simulation Type > Time Duration > Total Time	30 [s]
	Simulation Type > Time Steps > Timesteps ^{*†}	4*0.25, 2*0.5, 2*1.0, 13*2.0 [s]
	Simulation Type > Initial Time > Time	0 [s]

*. 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.

†. This list specifies 4 timesteps of 0.25 [s], then 2 timesteps of 0.5 [s], etc.

3. Click **OK**.

Modifying the Boundary Conditions

The only boundary condition which needs altering 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.

To Modify the Vent Inlet Boundary Condition

1. In the **Outline** workspace, expand the tree to `Simulation > CircVent > Vent`.
2. Right-click `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 main menu. 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*	FinalConcentration*abs(1-exp(-t/TimeConstant))

- *. 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 $(1 - \exp(-t/\text{TimeConstant}))$ 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 expressions have scheduled the concentration of smoke issued from the vent.

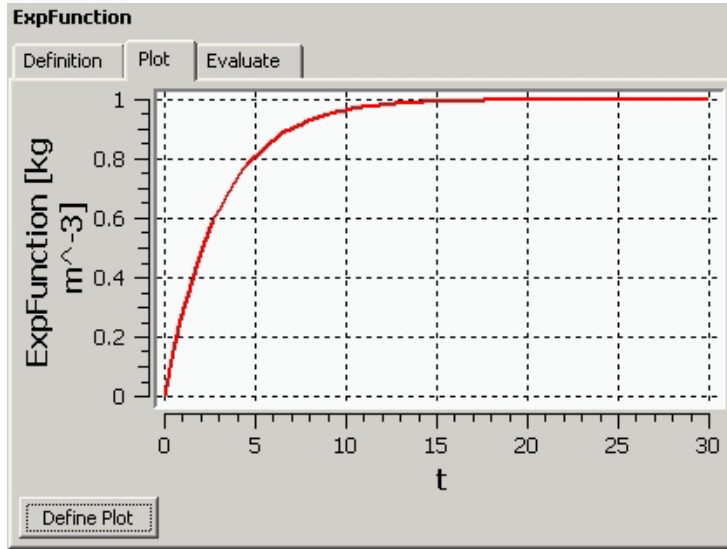
Plotting Smoke Concentration

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

Tab	Setting	Value
Plot	t	(Selected)
	Start of Range	0 [s]
	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 exponentially, and reaches 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 boundary `Vent`.

5. Apply the following settings

Tab	Setting	Value
Boundary Details	Additional Variables > smoke > Option	Value
	Additional Variables > smoke > Value*	ExpFunction


*. Click *Enter Expression*  to enter text.

6. Click **OK**.

Initialization Values

The steady state solution that you have finished calculating is used to supply the initial values to the ANSYS CFX-Solver. You can leave all of the initialization data set to **Automatic** and the initial values will be read automatically from the initial values file. Therefore, there is no need to revisit the initialization tab.


Modifying the Solver Control

1. Click *Solver Control* .
2. Set **Convergence Control** > **Max. Coeff. Loops** to 3.
3. Leave the other settings at their default values.
4. Click **OK** to set the solver control parameters.

Output Control


To allow results to be viewed at different timesteps, 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.

To Create Minimal Transient Results Files

1. From the main menu, select **Insert** > **Solver** > **Output Control**.
2. Click the **Trn Results** tab.
3. Click *Add new item*  and then click **OK** to accept the default name for the object. This creates a new transient results object. Each object can result in the production of many transient results files.
4. Apply the following settings to `Transient Results 1`

Setting	Value
Option	Selected Variables
Output Variables List*	Pressure, Velocity, smoke
Output Frequency > Option	Time List
Output Frequency > Time List [†]	1, 2, 3 [s]

*. 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.

†. 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. Create a second item with the default name `Transient Results 2` and apply the following settings to that item


Setting	Value
Option	Selected Variables
Output Variables List	Pressure, Velocity, smoke

Setting	Value
Output Frequency > Option	Time Interval
Output Frequency > Time Interval*	4 [s]

*. 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.

7. Click **OK**.

Writing the Solver (.def) File

1. Click *Write Solver File* .
2. Apply the following settings

Setting	Value
File name	CircVent.def
Quit CFX-Pre*	Select

*. If using ANSYS CFX-Pre in Standalone Mode.

3. Ensure **Start Solver Manager** is selected and click **Save**.
4. Quit ANSYS CFX-Pre, saving the simulation (.**cfx**) file at your discretion.


Obtaining a Solution to the Transient Problem

In this tutorial the ANSYS CFX-Solver will read the initial values for the problem from a file. For details, see [Initialization Values](#) (p. 103). You need to specify the file name.

Define Run will be displayed when the ANSYS CFX-Solver Manager launches. **Definition File** will already be set to the name of the definition file just written.

Notice that the text output generated by the ANSYS CFX-Solver 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 variable smoke is now plotted under the Additional Variables tab.

1. Under **Initial Values File**, click *Browse* .
2. Select **CircVentIni_001.res**, which is the results file of the steady-state problem with no smoke issuing from the chimney. If you have not run the first part of this tutorial, copy **CircVentIni_001.res** from the <CFXROOT>/examples/ directory to your working directory.
3. Click **Open**.
4. Click **Start Run**.
5. You may see a notice that the mesh from the initial values file will be used. This mesh is the same as in the definition file. Click **OK** to continue.

ANSYS 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.

6. When ANSYS CFX-Solver has finished, click **Yes** to post-process the results.
7. If using Standalone Mode, quit ANSYS CFX-Solver Manager.

Viewing the Results in ANSYS CFX-Post

In this tutorial, you will view the dispersion of smoke from the vent over time. When ANSYS CFX-Post is loaded, the results that are immediately available are those at the final timestep; in this case, at $t = 30$ s (this is nominally designated Final State).

Creating 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 $1 \text{ [m s}^{-1}\text{]}$. In this case, you are going to create an isosurface of `smoke` density (`smoke` is the additional variable that you specified earlier).

1. Right-click on 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 > Location > Isosurface** or under **Location**, click **Isosurface**.
3. Click **OK**.
4. Apply the following settings

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

5. Click **Apply**.
 - A bumpy surface will be displayed, showing the smoke starting to emerge 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 as the default settings on the **Color** tab were used.
 - When **Color Mode** is set to either `Constant` or `Use Plot Variable` for an isosurface, it appears as one color.
6. In **Geometry**, experiment by changing the **Value** 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 the **Value** to $0.002 \text{ [kg m}^{-3}\text{]}$.
8. Right-click on a blank spot in the viewer and select **Predefined Camera > Isometric View (Z up)**.

Viewing the Results at Different Timesteps


The **Timestep Selector** shows the **Time Step** (outer loop) number, the **Time Value** (simulated time in seconds) and the **Type** of results file that was saved at that timestep. You can see that **Partial** results files were saved (as requested in ANSYS CFX-Pre) for all timesteps 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 timestep 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 1 s.

Generating Output Files

You can produce image output from ANSYS CFX-Post.


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 main menu or click *Create text* .
2. Click **OK**.
3. In the **Text String** box, enter the following text.
Isosurface showing smoke concentration of 0.002 kg/m³ 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 **Appearance**, change **Color Mode** to *User Specified* and select a new color.
9. Click **Apply**.

JPEG output ANSYS CFX-Post can produce hard-copy output in several different forms. In the next section you will print in JPEG format.

1. Ensure a time value of 1 s is loaded.
2. Select **File > Print**, or click *Print* .
3. Under **Format** select *JPEG*.




4. Click *Browse*  next to the **File** data box.
5. Browse to the directory where you want the file saved.
6. Enter a name for the JPEG file.
7. Click **Save** to set the file name and directory.
This sets the path and name for the file.
8. To print to the file, click **Print**.
To view the file or make a hard copy, use an application that supports JPEG files.
9. Clear the visibility of the text object to hide it.

To Generate an MPEG File

You can generate an MPEG file to show the transient flow of the plume of smoke. To generate an MPEG file, you use the **Animation** dialog box in the same way as in Tutorial 1. However, to animate the plume of smoke, you need to animate over several timesteps.

Note: On the **Advanced** tab of **Animation Options**, there is a check box option called **Save frames as image files**. By selecting this option, the JPEG or PPM files used to encode each frame of the MPEG will persist after MPEG creation; otherwise, they are deleted.


Setting Keyframes

1. Click *Animation* .
2. Ensure that **Keyframe Animation** is selected.
3. Position the geometry so that you will be able to see the plume of smoke.
4. In the **Animation** dialog box, click *New*  to create `KeyFrameNo1`.
5. Load the time value of 30 s using the **Timestep Selector**.
6. Click *New*  in the **Animation** dialog box to create `KeyframeNo2`.

Defining additional options

During the production of a transient animation, various timesteps will be loaded and all objects will be updated to use the results from that timestep. Each frame of the animation must use one of the available timesteps.

In **Animation**, **Timestep** can be set to `Timestep Interpolation, TimeValue Interpolation Or Sequential Interpolation`. This setting affects which timestep is loaded for each frame.

1. Click *More Animation Options*  to show more animation settings.
2. Click **Options**.
3. Apply the following settings


Tab	Setting	Value
Options	Transient Case *	TimeValue Interpolation

*. This causes each frame to use the transient file having the closest time value.

4. Click **OK**.
5. Single click `KeyframeNo1`, then set **# of Frames** to 27 and press <Enter>.

The animation now contains a total of 29 frames (27 intermediate frames plus the two keyframes).

6. Select **Save MPEG**.


7. Click *Browse*  next to **Save MPEG**.


8. Under **File name**, type **CircVent .mpg**.

9. If required, set the path location to a different folder.

10. Click **Save**.

The MPEG file name (including path) is set. At this point, the animation has not yet been produced.

11. Click *To Beginning* .

12. Click *Play the animation* .

- The MPEG 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 MPEG file, you need to use a viewer that supports the MPEG format.

13. When you have finished, quit ANSYS CFX-Post.

Tutorial 5: Flow Around a Blunt Body

Introduction

This tutorial includes:

- [Tutorial 5 Features](#) (p. 109)
- [Overview of the Problem to Solve](#) (p. 111)
- [Defining a Simulation in ANSYS CFX-Pre](#) (p. 111)
- [Obtaining a Solution Using ANSYS CFX-Solver Manager](#) (p. 116)
- [Viewing the Results in ANSYS CFX-Post](#) (p. 119)

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

- [Setting the Working Directory](#) (p. 1)
- [Changing the Display Colors](#) (p. 2)

Unless you plan on running a session file, you should copy the sample files used in this tutorial from the installation folder for your software (<CFXROOT>/examples/) to your working directory. This prevents you from overwriting source files provided with your installation. If you plan to use a session file, please refer to [Playing a Session File](#) (p. 111).

Sample files referenced by this tutorial include:

- `BluntBody.pre`
- `BluntBodyDist.cse`
- `BluntBodyMesh.gtm`

Tutorial 5 Features

This tutorial addresses the following features of ANSYS CFX.

Component	Feature	Details	
ANSYS CFX-Pre	User Mode	General Mode	
	Simulation 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		
ANSYS CFX-Solver Manager	Parallel processing		
ANSYS CFX-Post	Plots	Default Locators	
		Outline Plot (Wireframe)	
		Sampling Plane	
		Streamline	
		Vector	
		Volume	
		Other	Changing the Color Range
	Instanting Transformation		
	Lighting Adjustment		
	Symmetry		
Viewing the Mesh			

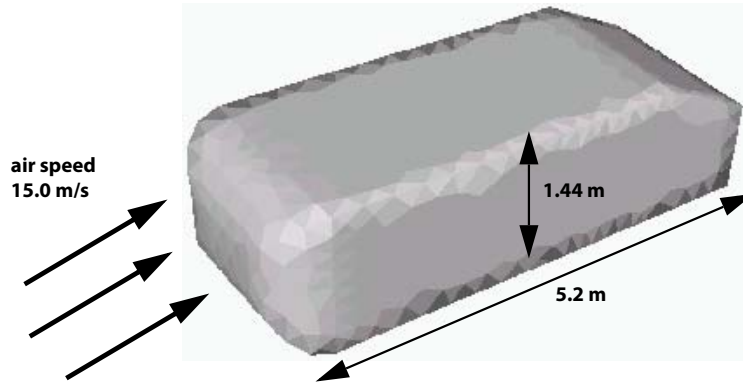
In this tutorial you will learn about:

- Solving and post-processing a case where the geometry has been omitted on one side of a symmetry plane.
- Using free slip wall boundaries on the sides of and above the domain 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 ANSYS CFX-Solver in parallel (optional).
- Creating vector plots in ANSYS CFX-Post with uniform spacing between the vectors.
- Creating a macro using power syntax in ANSYS CFX-Post.

Overview of the Problem to Solve

This example demonstrates external air flow over a generic vehicle 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.

Figure 1 External Air Flow Over a Generic Vehicle Body



Defining a Simulation in ANSYS CFX-Pre

The following sections describe the simulation setup in ANSYS CFX-Pre.

Playing a Session File

If you wish to skip past these instructions, and have ANSYS CFX-Pre set up the simulation automatically, you can select **Session > Play Tutorial** from the menu in ANSYS CFX-Pre, then run the session file: `BluntBody.pre`. After you have played the session file as described in earlier tutorials under [Playing the Session File and Starting ANSYS CFX-Solver Manager](#) (p. 87), proceed to [Obtaining a Solution Using ANSYS CFX-Solver Manager](#) (p. 116).

Creating a New Simulation

1. Start ANSYS CFX-Pre.
2. Select **File > New Simulation**.
3. Select **General** and click **OK**.
4. Select **File > Save Simulation As**.
5. Under **File name**, type `BluntBody`.
6. Click **Save**.

Importing the Mesh

1. Right-click `Mesh` and select **Import Mesh**.
2. Apply the following settings


Setting	Value
File name	BluntBodyMesh.gtm

3. Click **Open**.

Creating the Domain

The flow in the domain is expected to be turbulent and approximately isothermal. 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 an ideal gas as the fluid whereas previous tutorials have used a specific 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).

1. Click *Domain* , and set the name to `BluntBody`.
2. Apply the following settings to `BluntBody`:

Tab	Setting	Value
General Options	Basic Settings > Fluids List	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

3. Click **OK**.

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 condition 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 condition.

1. From the main menu, select **Insert > 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. In the region list, hold down the <Ctrl> key and select `Free1` and `Free2`.

- Click **OK**.

Creating the Boundary Conditions

The simulation requires inlet, outlet, wall (no slip and free slip) and symmetry plane boundary conditions. The regions for these boundary conditions were defined when the mesh was created (except for the composite region just created for the free slip wall boundary condition).

- Inlet Boundary**
- Click *Boundary Condition* .
 - Under **Name**, type `Inlet`.
 - 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 > Eddy Len. Scale	0.1 [m]

- Click **OK**.

- Outlet Boundary**
- Create a new boundary condition named `Outlet`.
 - 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]

- Click **OK**.

Free Slip Wall Boundary

The top and side surfaces of the rectangular region will use free slip wall boundary conditions.

- 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 is not an ideal boundary condition for this situation since the flow around the body will be affected by the close proximity to the walls. If this case was modeling a wind tunnel experiment, the domain should model the size and shape of the wind tunnel and use no-slip walls. If this case was modeling a blunt body open to the atmosphere, a much larger domain should be used to minimize the effect of the walls.

You will apply a single boundary condition to both walls by using the composite region defined earlier.

1. Create a new boundary condition named `FreeWalls`.
2. Apply the following settings:

Tab	Setting	Value
Basic Settings	Boundary Type	Wall
	Location	FreeWalls
Boundary Details	Wall Influence On Flow > Option	Free Slip

3. Click **OK**.

Symmetry Plane Boundary

1. Create a new boundary condition named `SymP`.
2. Apply the following settings:

Tab	Setting	Value
Basic Settings	Boundary Type	Symmetry
	Location	SymP

3. Click **OK**.

Wall Boundary on the Blunt Body Surface


1. Create a new boundary condition named `Body`.
2. Apply the following settings:

Tab	Setting	Value
Basic Settings	Boundary Type	Wall
	Location	Body
Boundary Details	Wall Influence On Flow > Option	No Slip

3. Click **OK**.

The remaining 2D regions (in this case, just the low Z face) will be assigned the default boundary condition which is an adiabatic, no-slip wall condition. In this case, the name of the default boundary condition is **Default Boundary**. Although the boundary conditions **Body** and **Default Boundary** are identical (except for their locations), the **Body** boundary condition was created so that, during post-processing, its location can be conveniently distinguished from the other adiabatic, no-slip wall surfaces.


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	15 [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 > Turbulence Eddy Dissipation	(Selected)

3. Click **OK**.


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]
	Convergence Criteria > Residual Target	1e-05

3. Click **OK**.

Writing the Solver (.def) File

1. Click *Write Solver File* .
2. Apply the following settings:

Setting	Value
File name	BluntBody.def
Quit CFX-Pre*	(Selected)

*. If using ANSYS CFX-Pre in Standalone Mode.

3. Ensure **Start Solver Manager** is selected and click **Save**.
4. If using Standalone Mode, quit ANSYS CFX-Pre, saving the simulation (**.cfx**) file at your discretion.

Obtaining a Solution Using ANSYS CFX-Solver Manager

This tutorial introduces the parallel solver capabilities of ANSYS 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 ANSYS CFX-Solver in parallel, proceed to [Obtaining a Solution in Serial](#) (p. 116).

If you do not know if you have a license to run the ANSYS 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. 116).

If you would like to solve this tutorial in parallel on the same machine, proceed to [Obtaining a Solution with Local Parallel](#) (p. 117).

If you would like to solve this tutorial in parallel across different machines, proceed to [Obtaining a Solution with Distributed Parallel](#) (p. 117).

Obtaining a Solution in Serial

When ANSYS CFX-Pre has shut down and ANSYS CFX-Solver Manager has started, you can obtain a solution to the CFD problem by using the following procedure.

1. Click **Start Run**.
2. Click **Yes** to process the results in ANSYS CFX-Post.
3. If using Standalone Mode, quit ANSYS CFX-Solver Manager.
Continue this tutorial from [Viewing the Results in ANSYS CFX-Post](#) (p. 119).

Obtaining a Solution in Parallel

Background to Parallel Running in ANSYS CFX

Using the parallel capability of the ANSYS 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 ANSYS CFX-Solver to calculate the solution:

- Your mesh will be divided into the number of partitions that you have chosen.
- The ANSYS CFX-Solver runs separately on each of the partitions on the selected machine(s).
- The results that one ANSYS CFX-Solver process calculates affects the other ANSYS CFX-Solver processes at the interface between the different sections of the mesh.
- All of the ANSYS 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 ANSYS 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.

Obtaining a Solution with Local Parallel

To run in local parallel mode, the machine you are on must have more than one processor. In ANSYS 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 ANSYS CFX-Solver afterwards.
2. Set **Run Mode** to `PVM 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 ANSYS 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. Click *Post-Process Results* .
9. If using ANSYS CFX-Solver in Standalone Mode, select **Shut down Solver Manager**, and then click **OK**.

Continue this tutorial from [Text Output when Running in Parallel](#) (p. 118).

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 ANSYS 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 ANSYS CFX-Solver afterwards.
2. Set **Run Mode** to `PVM 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 ANSYS 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 ANSYS 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. Click **Yes** to post-process the results when the completion message appears at the end of the run.

13. Close ANSYS CFX-Solver Manager.

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:   Wed Nov 28 15:18:40 2005
```

This tells you that the information following is concerned with the partitioning. After the partitioning job has finished, you will find:

```
CPU-Time requirements:
- Preparations                               1.460E+00 seconds
- Low-level mesh partitioning                 1.000E-01 seconds
- Global partitioning information             3.100E-01 seconds
- Vertex, element and face partitioning information 1.600E-01 seconds
- Element and face set partitioning information 5.000E-02 seconds
- Summed CPU-time for mesh partitioning      2.080E+00 seconds
+-----+
|                                     Job Information                                     |
+-----+
Host computer:  fastmachine1
Job finished:   Wed Nov 28 15:19:16 2005
Total CPU time: 1.143E+01 seconds
                or: (           0:           0:           0:    11.428 )
                  (    Days:    Hours:    Minutes:    Seconds )
```

This marks the end of the partitioning job. The ANSYS CFX-Solver now begins to solve your parallel run:

```
+-----+
|                                     Job Information                                     |
+-----+
Run mode:      parallel run (PVM)
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 ANSYS CFX-Post later on in the tutorial.

When the ANSYS CFX-Solver finishes, the output file displays the job information and a pop-up message to indicate completion of the run.

Viewing the Results in ANSYS CFX-Post

In this tutorial, a vector plot is created in ANSYS CFX-Post. This will let you see how the flow behaves around the body. You will also use symmetry planes and learn more about manipulating the geometry view in the viewer.

Using Symmetry Planes

Earlier in this tutorial you used a symmetry plane boundary condition 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. ANSYS CFX-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.

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 ANSYS CFX-Post features that you have used in earlier tutorials will not be described in detail. New features will be described in detail.

1. Right-click a blank area in the viewer and select **Predefined Camera > View Towards +X**.

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. 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
Render	Draw Faces	(cleared)

3. Click **Apply**.
This creates a plane in the same location as the symmetry plane defined in ANSYS 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**.

Using the Reflection Transform

You can use the transform when creating or editing graphics objects. For example, you can modify the Wireframe view to use it 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.

Creating 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.

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 Towards +Y**.
This ensures that the changes can be seen.
2. Create a new plane named `Sample`.
3. Apply the following settings:

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	Draw Faces	(Cleared)
	Draw 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. Hide the plane by clearing the visibility check box next to `Sample`.

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.

- 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

- Click **Apply**.
- Change the location of the Vector plot by applying the following setting:

Tab	Setting	Value
Geometry	Definition > Locations	SymP

- Click **Apply**.

Creating a Pressure Plot

- Apply the following settings to the boundary condition named `Body`:

Tab	Setting	Value
Color	Mode	Variable
	Variable	Pressure
View	Apply Instancing Transform > Transform	Instance Transform 1

- Click **Apply**.
- Apply the following settings to `SymP`:

Tab	Setting	Value
Render	Draw Faces	(Cleared)
	Draw Line	(Selected)


- 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.

Creating Surface Streamlines

In order to show the path of air along the surface of the blunt body, surface streamlines can be made as follows:

- Clear the visibility of `Body`, `SymP` and `Vector 1`.
- Create a new plane named `Starter`.
- Apply the following settings

Tab	Setting	Value
Geometry	Definition > Method	YZ Plane
	X	-0.1 [m]


- Click **Apply**.
The plane appears just upstream of the blunt body.
- Clear the visibility check box for the plane.
This hides the plane from view, although the plane still exists.
- Click **Streamline**  and click **OK** to accept the default name.
- 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

- Apply** the following settings.
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.

Moving Objects

In ANSYS CFX-Post, you can reposition some locator objects directly in the viewer by using the mouse.

- Select the visibility check box for the plane named *Starter*.
- Select the  **Single Select** mouse pointer from the **Selection Tools** toolbar.
- 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.

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.

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. Clear the visibility of all previous plots.
2. Under the **Outline** tab, apply the following settings to `BluntBodyDefault`:

Tab	Setting	Value
Color	Mode	Variable
	Variable	$Yplus^*$
View	Apply Instancing Transform > Transform	Instance Transform 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	$Yplus^*$
View	Apply Instancing Transform > Transform	Instance Transform 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**.

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 ANSYS CFX-Post.

Synopsis

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 ANSYS CFX-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 ANSYS CFX-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.

Procedure

1. Play the session file named **BluntBodyDist.cse**.
2. Right-click a blank area in the viewer and select **Predefined Camera > View Towards -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.

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 ANSYS CFX-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 ANSYS CFX-Post.

Tutorial 6: Buoyant Flow in a Partitioned Cavity

Introduction

This tutorial includes:

- [Tutorial 6 Features](#) (p. 128)
- [Overview of the Problem to Solve](#) (p. 128)
- [Defining a Simulation in ANSYS CFX-Pre](#) (p. 129)
- [Obtaining a Solution using ANSYS CFX-Solver Manager](#) (p. 134)
- [Viewing the Results in ANSYS CFX-Post](#) (p. 135)

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

- [Setting the Working Directory](#) (p. 1)
- [Changing the Display Colors](#) (p. 2)

Unless you plan on running a session file, you should copy the sample files used in this tutorial from the installation folder for your software (`<CFXROOT>/examples/`) to your working directory. This prevents you from overwriting source files provided with your installation. If you plan to use a session file, please refer to [Playing a Session File](#) (p. 129).

Sample files referenced by this tutorial include:

- **Buoyancy2D.geo**
- **Buoyancy2D.pre**

Tutorial 6 Features

This tutorial addresses the following features of ANSYS CFX.

Component	Feature	Details	
ANSYS CFX-Pre	User Mode	General Mode	
	Simulation 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 Example		
Transient Results File			
ANSYS CFX-Post	Plots	Default Locators	
	Report		
	Other		Time Step Selection
			Transient Animation

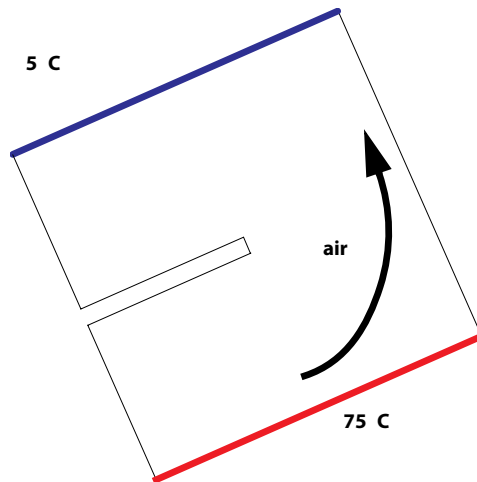
In this tutorial you will learn about:

- Using CFX-4 Mesh Import.
- Setting up a time dependent (transient) simulation.
- Modeling buoyant flow.

Overview of the Problem to Solve

This tutorial demonstrates the capability of ANSYS CFX in modeling buoyancy-driven flows which require the inclusion of gravitational effects.

The model is a 2D partitioned cavity containing air. 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. A transient simulation is set up 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.



The mesh for the cavity was created in CFX-4 and has been provided.

Defining a Simulation in ANSYS CFX-Pre

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.

Playing a Session File

If you wish to skip past these instructions, and have ANSYS CFX-Pre set up the simulation automatically, you can select **Session > Play Tutorial** from the menu in ANSYS CFX-Pre, then run the session file: **Buoyancy2D.pre**. After you have played the session file as described in earlier tutorials under [Playing the Session File and Starting ANSYS CFX-Solver Manager](#) (p. 87), proceed to [Obtaining a Solution using ANSYS CFX-Solver Manager](#) (p. 134).

Creating a New Simulation

1. Start ANSYS CFX-Pre.
2. Create a new simulation using **General Mode**.
3. Select **File > Save Simulation As** and set **File name** to `Buoyancy2D`.
4. Click **Save**.

Importing the Mesh

1. Right-click `Mesh` and select **Import Mesh**. The **Import Mesh** dialog box appears.

- Apply the following settings

Setting	Value
File type	CFX-4
File name	Buoyancy2D.geo*


*. This file is in your tutorial directory.

- Click **Open**.

Simulation Type

The default units and coordinate frame settings are suitable for this tutorial, but the simulation 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 in the later part of the tutorial.

- Click *Simulation Type* .
- Apply the following settings

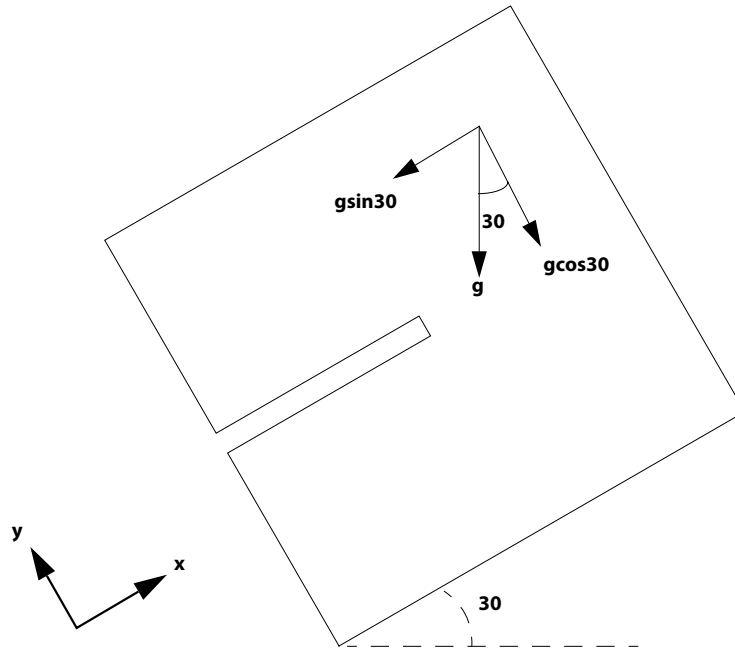
Tab	Setting	Value
Basic Settings	Simulation Type > Option	Transient
	Simulation Type > Time Duration > Total Time*	2 [s]
	Simulation Type > Time Steps > Timesteps†	0.025 [s]
	Simulation Type > Initial Time > Time	0 [s]

*. This is the total duration, in real time, for the simulation

†. This is the interval from one step, in real time, to the next. The simulation will continue, moving forward in time by 0.025 s, until the total time has been reached


- Click **OK**.

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.


To Create a New Domain

1. Click *Domain* , and set the name to `Buoyancy2D`.
2. Apply the following settings to `Buoyancy2D`

Tab	Setting	Value
General Options	Basic Settings > Fluids List	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] [*]
	Domain Models > Buoyancy > Buoy. Ref. Temp.	40 [C] [†]
Fluid Models	Heat Transfer > Option	Thermal Energy
	Turbulence > Option	None (Laminar)

*. This produces a gravity vector which simulates the tilt of the cavity

†. Do not forget to change the units. 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.

3. Click **OK**.

Creating the Boundary Conditions

Hot and Cold Wall Boundary

You will create a wall boundary condition with a fixed temperature of 75 C on the bottom surface of the cavity, as follows:

1. Create a new boundary condition 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 condition 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**.

Symmetry Plane Boundary

A single symmetry plane boundary condition can be used for the front and back of the cavity.

1. Create a new boundary condition named `SymP`.
2. Apply the following settings

Tab	Setting	Value
Basic Settings	Boundary Type	Symmetry
	Location	SYMMET1, SYMMET2*


*. Use the <Ctrl> key to select more than one region.

3. Click **OK**.

The default adiabatic wall boundary condition will automatically be applied to the remaining boundaries.

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 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**.

Setting Output Control


1. Click *Output Control* .
2. Click the **Trn Results** tab.
3. Create a new Transient Results item with the default name.
4. Apply the following settings

Tab	Setting	Value
Trn Results	Transient Results > Transient Results 1 > Option	Selected Variables
	Transient Results > Transient Results 1 > Output Variables List*	Pressure, Temperature, Velocity
	Transient Results > Transient Results 1 > Output Frequency > Option	Time Interval
	Transient Results > Transient Results 1 > Output Frequency > Time Interval	0.1 [s]

*. 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.

5. Click **OK**.

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
	Convergence Criteria > Residual Type	RMS
	Convergence Criteria > Residual Target	1.E-4*

*. 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**.

Writing the Solver (.def) File

1. Click *Write Solver File* .
2. Apply the following settings:

Setting	Value
File name	Buoyancy2D.def
Quit CFX-Pre*	(Selected)

*. If using ANSYS CFX-Pre in Standalone Mode.

3. Ensure **Start Solver Manager** is selected and click **Save**.
4. If using Standalone Mode, quit ANSYS CFX-Pre, saving the simulation (.cfx) file at your discretion.

Obtaining a Solution using ANSYS CFX-Solver Manager

When ANSYS CFX-Pre has shut down and ANSYS 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 ANSYS 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. Click **Yes** to post-process the results when the completion message appears at the end of the run.
3. If using Standalone Mode, quit ANSYS CFX-Solver Manager.

Viewing the Results in ANSYS CFX-Post

In this section, you will create a report in ANSYS CFX-Post. You will also make an animation to see changes in temperature with time.

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.

Plots

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

Contour Plot

1. Click the **3D Viewer** tab and right-click a blank area of the viewer, then select **Predefined Camera > View Towards -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 ANSYS CFX-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.

Point Locators


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.

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.`

Figure

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.

Time Chart

1. Select **Insert > Chart** from the main menu.
2. Accept the default name by clicking **OK**.
3. Set **Title** to `Temperature versus Time`.
4. Set **Type** to `Time`.
5. Click the **Chart Line 1** tab.
6. Set **Line Name** to `Temperature at Point 1`.
7. Set **Method** to `Point`.
8. Set **Location** to `Point 1`.
9. Set **Time Variable > Variable** to `Temperature`.
10. Click **Apply**.
A chart object will appear under the `Report` branch in the tree view. The chart itself will appear in the **Chart Viewer** tab. 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.
11. Click **New Line** (on the **Chart Line 1** tab).
12. Set **Line Name** to `Temperature at Point 2`.
13. Set **Location** to `Point 2` and **Time Variable > Variable** to `Temperature`.
14. Click **Apply**.
A second chart line will appear in the chart, representing the temperature at `Point 2`.

Table

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 ANSYS CFX-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**.

Customized Report

Right-click the `Report` object and select **Refresh** 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.

Animations

Use the animation feature to see the changing temperature field. The animation feature was used in [Tutorial 4: Flow from a Circular Vent](#) (p. 93).

Completion

When you have finished, quit ANSYS CFX-Post.

Tutorial 7: Free Surface Flow Over a Bump

Introduction

This tutorial includes:

- [Tutorial 7 Features](#) (p. 139)
- [Overview of the Problem to Solve](#) (p. 140)
- [Defining a Simulation in ANSYS CFX-Pre](#) (p. 141)
- [Obtaining a Solution using ANSYS CFX-Solver Manager](#) (p. 148)
- [Viewing the Results in ANSYS CFX-Post](#) (p. 149)
- [Using a Supercritical Outlet Condition](#) (p. 154)

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

- [Setting the Working Directory](#) (p. 1)
- [Changing the Display Colors](#) (p. 2)

Unless you plan on running a session file, you should copy the sample files used in this tutorial from the installation folder for your software (<CFXROOT>/examples/) to your working directory. This prevents you from overwriting source files provided with your installation. If you plan to use a session file, please refer to [Playing a Session File](#) (p. 141).

Sample files referenced by this tutorial include:

- **Bump2D.pre**
- **Bump2DExpressions.ccl**
- **Bump2Dpatran.out**

Tutorial 7 Features

This tutorial addresses the following features of ANSYS CFX:

Component	Feature	Details	
ANSYS CFX-Pre	User Mode	General Mode	
	Simulation Type	Steady State	
	Fluid Type	General Fluid	
	Domain Type	Single Domain	
	Turbulence Model	k-Epsilon	
	Heat Transfer	None	
	Buoyant Flow		
	Multiphase		
	Boundary Conditions		Inlet (Subsonic)
			Outlet (Subsonic)
			Symmetry Plane
			Wall: No-Slip
			Wall: Free-Slip
	CEL (CFX Expression Language)		
Mesh Adaption			
Timestep	Physical Time Scale		
ANSYS CFX-Post	Plots	Default Locators	
		Isosurface	
		Polyline	
		Sampling Plane	
		Vector	
		Volume	
	Other	Chart Creation	
		Title/Text	
	Viewing the Mesh		

In this tutorial you will learn about:

- Mesh import in PATRAN Neutral format.
- Setting up a 2D problem.
- Setting 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.)
- Mesh adaption to refine the mesh where the volume fraction gradient is greatest. (This aids in the development of a sharp interface between the liquid and gas.)

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 semi-circular bump of radius 30 mm. The flow upstream of the bump is subcritical. The downstream conditions are not known but can be estimated using an analytical 1D calculation or data tables for flow over a bump.

Defining a Simulation in ANSYS CFX-Pre

The following sections describe the simulation setup in ANSYS CFX-Pre.

Playing a Session File

If you wish to skip past these instructions, and have ANSYS CFX-Pre set up the simulation automatically, you can select **Session > Play Tutorial** from the menu in ANSYS CFX-Pre, then run the session file: **Bump2D.pre**. After you have played the session file as described in earlier tutorials under [Playing the Session File and Starting ANSYS CFX-Solver Manager](#) (p. 87), proceed to [Obtaining a Solution using ANSYS CFX-Solver Manager](#) (p. 148).

Creating a New Simulation

1. Start ANSYS CFX-Pre.
2. Select **File > New Simulation**.
3. Select **General** and click **OK**.
4. Select **File > Save Simulation As**.
5. Under **File name**, type `Bump2D`.
6. Click **Save**.


Importing the Mesh

1. Right-click `Mesh` and select **Import Mesh**. The **Import Mesh** dialog box appears.
2. Apply the following settings

Setting	Value
File type	PATRAN Neutral
File name	Bump2Dpatran.out

3. Click **Open**.
4. Right-click a blank area in the viewer and select **Predefined Camera > View Towards -Z** from the shortcut menu.

Viewing the Region Labels

1. Click *Label and Marker Visibility* .
2. Apply the following settings

Tab	Setting	Value
Label Options	Show Labels	(Selected)
	Show Labels > Show Primitive3D Labels	(Selected)
	Show Labels > Show Primitive2D Labels	(Selected)

3. Click **OK**.

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. 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 post-processing 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 import expressions from a file.

- [Creating Expressions](#) (p. 142)
- [Reading Expressions From a File](#) (p. 143)

Creating Expressions

1. Right-click **Expressions** in the tree view and select **Insert > Expression**.
2. Set the name to `UpH` and click **OK**.
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 density of the fluid, 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]
DenH	998 [kg m ⁻³]
UpVFAir	step((y-UpH)/1[m])
UpVFWater	1-UpVFAir
UpPres	DenH*g*UpVFWater*(UpH-y)
DownVFAir	step((y-DownH)/1[m])
DownVFWater	1-DownVFAir
DownPres	DenH*g*DownVFWater*(DownH-y)

5. Proceed to [Creating the Domain](#) (p. 143).


Reading Expressions From a File

1. Copy the file `Bump2DExpressions.ccl` to your working directory from the ANSYS CFX examples directory.
2. Select **File > Import CCL**.
3. When **Import CCL** appears, ensure that **Append** is selected.
4. Select `Bump2DExpressions.ccl`.
5. Click **Open**.
6. After the file has been imported, use the **Expression** tree view to view the expressions that have been created.

Creating the Domain

1. Right click `Simulation` in the **Outline** tree view and ensure that **Automatic Default Domain** is selected. 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
General Options	Basic Settings > Fluids List	Air at 25 C, 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.*	-g
	Domain Models > Buoyancy > Gravity Z Dirn.	0 [m s ⁻²]
	Domain Models > Buoyancy > Buoy. Ref. Density [†]	1.185 [kg m ⁻³]
	Domain Models > Buoyancy > Ref Location > Option	Automatic
Fluid Models	Multiphase Options > Homogeneous Model [‡]	(Selected)
	Multiphase Options > Free Surface Model > Option	Standard
	Heat Transfer > Option	Isothermal
	Heat Transfer > Fluid Temperature	25 C
	Turbulence > Option	k-Epsilon

- *. You need to click *Enter Expression*  beside the field first.
- †. Always set Buoyancy Reference Density to the density of the least dense fluid in free surface calculations.
- ‡. The homogeneous model solves for a single solution field. This is only appropriate in some simulations.

3. Click **OK**.

Creating the Boundary Conditions

Inlet Boundary

1. Create a new boundary condition 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 > Option > Normal Speed	0.26 [m s ⁻¹]
	Turbulence > Option	Intensity and Length Scale
	Turbulence > Value	0.05
	Turbulence > Eddy Len. Scale*	UpH
Fluid Values	Boundary Conditions	Air as 25 C
	Air at 25 C > Volume Fraction > Volume Fraction	UpVFAir
	Boundary Conditions	Water
	Water > Volume Fraction > Volume Fraction	UpVFWater

*. Click the Enter Expression icon.

3. Click **OK**.

Outlet Boundary

1. Create a new boundary condition 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**.

Symmetry Boundary

1. Create a new boundary condition named `front`.
2. Apply the following settings

Tab	Setting	Value
Basic Settings	Boundary Type	Symmetry
	Location	FRONT

3. Click **OK**.
4. Create a new boundary condition named `back`.
5. Apply the following settings

Tab	Setting	Value
Basic Settings	Boundary Type	Symmetry
	Location	BACK

6. Click **OK**.

Wall and Opening Boundaries

1. Create a new boundary condition named `top`.
2. Apply the following settings


Tab	Setting	Value
Basic Settings	Boundary Type	Opening
	Location	TOP
Boundary Details	Mass And Momentum > Option	Static Pres. (Entrain)
	Mass And Momentum > Relative Pressure	0 [Pa]
	Turbulence > Option	Zero Gradient
Fluid Values	Boundary Conditions	Air at 25 C
	Boundary Conditions > Air at 25 C > 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 condition named `bottom`.
5. Apply the following settings

Tab	Setting	Value
Basic Settings	Boundary Type	Wall
	Location	BOTTOM1, BOTTOM2, BOTTOM3
Boundary Details	Wall Influence on Flow > Option	No Slip
	Wall Roughness > Option	Smooth Wall

6. Click **OK**.


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	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
	Initial Conditions > Turbulence Eddy Dissipation	(Selected)
Fluid Settings	Fluid Specific Initialization > Air at 25 C	(Selected)
	Air at 25 C > Initial Conditions > Volume Fraction > Option	Automatic with Value
	Air at 25 C > Initial Conditions > Volume Fraction > Volume Fraction	UpVFAir
Fluid Settings	Fluid Specific Initialization > Water	(Selected)
	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**.

Setting Mesh Adaption Parameters


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 at 25 C.Volume Fraction
	Adaption Criteria > Max. Num. Steps	2
	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

3. Click **OK**.

Setting Solver Control

Important: Setting **Max Iterations** to 200 and **Number of Adaption Levels** to 2 with a maximum of 100 timesteps each, results in a total maximum number of timesteps of 400 ($2 \times 100 + 200 = 400$).

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]
Advanced Options	Multiphase Control	(Selected)
	Multiphase Control > Volume Fraction Coupling	(Selected)
	Multiphase Control > Volume Fraction Coupling > Option	Coupled

Note: The options selected above activate the Coupled Volume Fraction solution algorithm. This algorithm typically converges better than the Segregated Volume Fraction algorithm for buoyancy-driven problems such as this tutorial, which requires a 0.05 [s] timescale using the Segregated Volume Fraction algorithm compared with 0.25 [s] for the Coupled Volume Fraction algorithm.

Note:

3. Click **OK**.

Writing the Solver (.def) File

1. Click *Write Solver File* .
2. Apply the following settings:

Setting	Value
File name	Bump2D.def
Quit CFX-Pre *	(Selected)

*. If using ANSYS CFX-Pre in Standalone Mode.

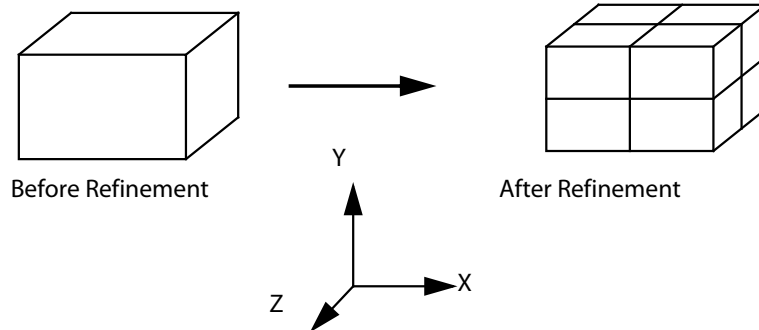
3. Ensure **Start Solver Manager** is selected and click **Save**.
4. If using Standalone Mode, quit ANSYS CFX-Pre, saving the simulation (**.cfx**) file at your discretion.

Obtaining a Solution using ANSYS CFX-Solver Manager

When ANSYS CFX-Pre has shut down and ANSYS CFX-Solver Manager has started, the solution will be obtained.

Within 100 iterations, the first adaption step will be performed. Information will be written to the OUT file, containing the number of elements refined and the size of the new mesh.

After mesh refinement, there will be a jump in the residual levels. This is because the solution from the old mesh is interpolated on to the new mesh. A new residual plot will also appear 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 has been achieved. It is common for convergence in a residual sense to be difficult to obtain in a free surface simulation. This is 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 for a few timesteps, starting from a results file that you think is converged, or by writing some backup results files 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. Click **Start Run**.
2. Click **Yes** to post-process the results when the completion message appears at the end of the run.
3. If using Standalone Mode, quit ANSYS CFX-Solver Manager.

Viewing the Results in ANSYS CFX-Post

1. Select **View Towards -Z** by right-clicking on a blank area in the viewer and selecting **Predefined Camera > View Towards -Z**.
2. Zoom in so the geometry fills the Viewer.
3. In the tree view under `Bump2D`, edit `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`.

Creating Velocity Vector Plots

The next step involves creating a sampling plane to display velocity vectors for `Water`.

1. 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]
	Plane Bounds > Y Size	0.3 [m]
	Plane Bounds > X Angle	0 [degree]
	Plane Type	Sample
	X Samples	160
	Y Samples	40
Render	Draw Faces	(Cleared)
	Draw Lines	(Selected)

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*	Water.Velocity
Symbol	Symbol Size	0.5

- *. Since fluids in a free-surface calculation share the same velocity field, only the velocity of the first non-vapour fluid is available. The other allowed velocities are superficial velocities. For details, see [Further Post-processing](#) (p. 154).

7. Click **Apply**.
8. Apply the following settings

Tab	Setting	Value
Geometry	Definition > Variable	Air at 25 C.Superficial Velocity
Symbol	Symbol Size	0.15
	Normalize Symbols	(Selected)

9. Click **Apply**.

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.

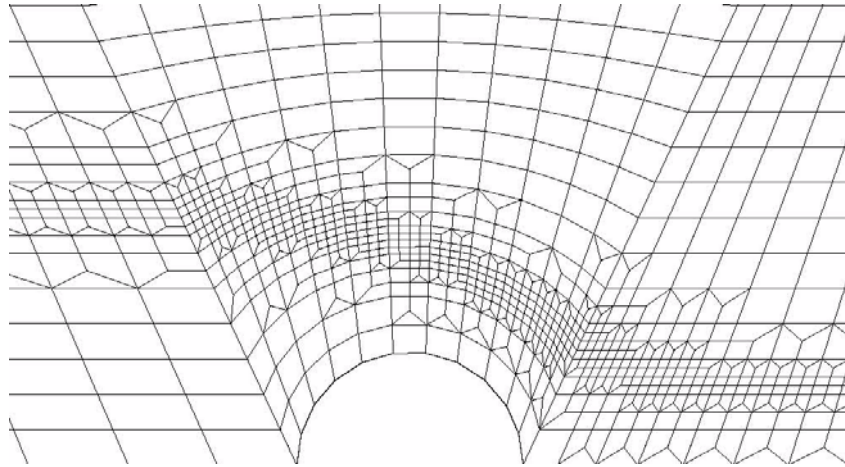
1. Clear the check box next to `Vector 1..`
2. Zoom in so the geometry fills the Viewer.
3. In **Outline** under `Default Domain`, edit `front`.
4. Apply the following settings

Tab	Setting	Value
Color	Mode	Constant
Render	Draw Faces	(Cleared)
	Draw 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 since it is not possible to recreate hexahedral elements in ANSYS 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, resulting in poor mesh quality.

Figure 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
	Definition > Mode	At Value
	Definition > Value	1
Render	Draw Faces	(Cleared)
	Draw Lines	(Selected)
	Draw Lines > Line Width	2
	Draw Lines > Color Mode	User Specified
	Draw Lines > Line Color	(Green)

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

Tab	Setting	Value
Render	Draw Faces	(Selected)
	Draw Lines	(Selected)
	Draw Lines > Line Width	4
	Draw Lines > Color Mode	User Specified
	Draw Lines > Line Color	(Black)

11. Click **Apply**.

You will see a band of white (with black lines) which 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 (since 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 > Location	Bump2D
	Definition > Variable*	(Any Vector Variable)
Color	Mode	Variable
	Variable	Refinement Level
Symbol	Symbol	Cube
	Symbol Size	0.02
	Normalize Symbols	(Selected)

*. The variable's magnitude and direction do not matter since you will change the vector symbol to a cube with a normalized size.

15. Click **Apply**.

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.

Creating a Chart

Next, you will create a chart to show how the height of the free surface varies along the length of the channel. To do this, you will need a Polyline which follows the free surface. You can create the Polyline from the intersecting line between one of the Symmetry planes and an Isosurface which shows the free surface. First you must create the Isosurface.

1. Clear the visibility check boxes for all of the 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

- Click **Apply**.
Creating isosurfaces using this method is a good way to visualize a free surface in a 3D simulation.
- Right-click any blank area in the viewer, select **Predefined Camera**, then select **Isometric View (Y up)**.

Creating a Polyline to Follow the Free Surface

These steps explain creating a Polyline which follows the free surface:

- Clear the visibility check box for `Isosurface 1`.
- Create a new polyline named `Polyline 1`.
- Apply the following settings

Tab	Setting	Value
Geometry	Method	Boundary Intersection
	Boundary List	front
	Intersect With	Isosurface 1

- Click **Apply**.
A green line is displayed that follows the high-Z edge of the isosurface.

Creating a Chart to Show the Height of the Surface

- Create a new chart named `Chart 1`.
The **Chart Viewer** tab is selected.
- Apply the following settings

Tab	Setting	Value
Chart Line 1	Line Name	free surface height
	Location	Polyline 1
	X Axis > Variable	X
	Y Axis > Variable	Y
	Appearance > Symbols	Rectangle
Chart	Title	Free Surface Height for Flow over a Bump

- Click **Apply**.
As discussed in [Creating Expressions for Initial and Boundary Conditions](#) (p. 142), an approximate outlet elevation is imposed as part of the boundary condition, 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, since 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.

Further Post-processing

You may wish 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.

Using a Supercritical Outlet Condition

For supercritical free surface flows, the supercritical outlet boundary condition is usually the most appropriate boundary condition for the outlet, since 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 condition 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.

Tutorial 8:

Supersonic Flow Over a Wing

Introduction

This tutorial includes:

- [Tutorial 8 Features](#) (p. 155)
- [Overview of the Problem to Solve](#) (p. 157)
- [Defining a Simulation in ANSYS CFX-Pre](#) (p. 157)
- [Obtaining a Solution using ANSYS CFX-Solver Manager](#) (p. 162)
- [Viewing the Results in ANSYS CFX-Post](#) (p. 162)

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

- [Setting the Working Directory](#) (p. 1)
- [Changing the Display Colors](#) (p. 2)

Unless you plan on running a session file, you should copy the sample files used in this tutorial from the installation folder for your software (<CFXROOT>/examples/) to your working directory. This prevents you from overwriting source files provided with your installation. If you plan to use a session file, please refer to [Playing a Session File](#) (p. 157).

Sample files referenced by this tutorial include:

- `WingSPS.pre`
- `WingSPSMesh.out`

Tutorial 8 Features

This tutorial addresses the following features of ANSYS CFX.

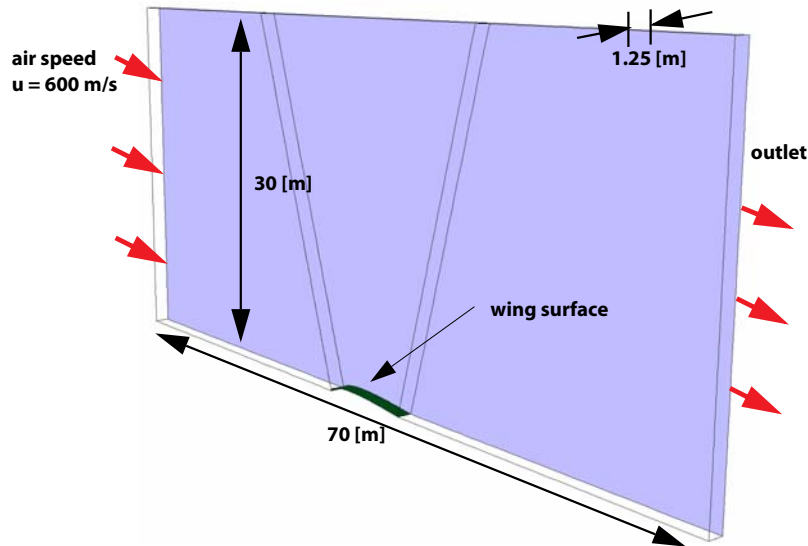
Component	Feature	Details
ANSYS CFX-Pre	User Mode	General Mode
	Simulation Type	Steady State
	Fluid Type	Ideal Gas
	Domain Type	Single Domain
	Turbulence Model	Shear Stress Transport
	Heat Transfer	Total Energy
	Boundary Conditions	Inlet (Supersonic)
		Outlet (Supersonic)
		Symmetry Plane
		Wall: No-Slip
		Wall: Adiabatic
Wall: Free-Slip		
Domain Interfaces	Fluid-Fluid (No Frame Change)	
Timestep	Auto Time Scale	
ANSYS CFX-Post	Plots	Contour
		Default Locators
		Vector
	Other	Variable Details View

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 the wing surface.
- Defining custom vector variables for use in visualizing pressure distribution.

Overview of the Problem to Solve

This example demonstrates the use of ANSYS 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 is imported into ANSYS CFX-Pre.



Defining a Simulation in ANSYS CFX-Pre

The following sections describe the simulation setup in ANSYS CFX-Pre.

Playing a Session File

If you wish to skip past these instructions, and have ANSYS CFX-Pre set up the simulation automatically, you can select **Session > Play Tutorial** from the menu in ANSYS CFX-Pre, then run the session file: `wingSPS.pre`. After you have played the session file as described in earlier tutorials under [Playing the Session File and Starting ANSYS CFX-Solver Manager](#) (p. 87), proceed to [Obtaining a Solution using ANSYS CFX-Solver Manager](#) (p. 162).

Creating a New Simulation

1. Start ANSYS CFX-Pre.
2. Select **File > New Simulation**.
3. Select **General** and click **OK**.
4. Select **File > Save Simulation As**.
5. Under **File name**, type `WingSPS`.
6. Click **Save**.

Importing the Mesh

1. Right-click `Mesh` and select **Import Mesh**. The **Import Mesh** dialog box appears.
2. Apply the following settings

Setting	Value
File type	PATRAN Neutral
File name	WingSPSMesh.out

3. Click **Open**.
4. Right-click a blank area in the viewer and select **Predefined Camera > Isometric View (Y up)** from the shortcut menu.

Creating the Domain

Creating a New Domain

1. Right click `Simulation` in the **Outline** tree view and ensure that **Automatic Default Domain** is selected. A domain named `Default Domain` should now appear under the `Simulation` branch.
2. Double click it and apply the following settings

Tab	Setting	Value
General Options	Basic Settings > Location	WING
	Fluids List	Air Ideal Gas
	Domain Models > Pressure > Reference Pressure*	1 [atm]
Fluid Models	Heat Transfer > Option	Total Energy [†]
	Turbulence > Option	Shear Stress Transport

- *. When using an ideal gas, it is important to set an appropriate reference pressure since some properties depend on the absolute pressure level.
- †. The Total Energy model is appropriate for high speed flows since it includes kinetic energy effects.

3. Click **OK**.

Creating the Boundary Conditions

Inlet Boundary

1. Create a new boundary condition named `Inlet`.
2. Apply the following settings

Tab	Setting	Value
Basic Settings	Boundary Type	Inlet
	Location	INLET

Tab	Setting	Value
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 > Value	0.01
	Turbulence > Eddy Len. Scale	0.02 [m]
	Heat Transfer > Static Temperature	300 [K]

3. Click **OK**.

Outlet Boundary

1. Create a new boundary condition 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**.

Symmetry Plane Boundary

1. Create a new boundary condition named `SymP1`.
2. Apply the following settings

Tab	Setting	Value
Basic Settings	Boundary Type	Symmetry
	Location	SIDE1

3. Click **OK**.
4. Create a new boundary condition 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 condition named `Bottom`.
8. Apply the following settings

Tab	Setting	Value
Basic Settings	Boundary Type	Symmetry
	Location	BOTTOM

9. Click **OK**.

Free Slip Boundary

1. Create a new boundary condition named `Top`.
2. Apply the following settings


Tab	Setting	Value
Basic Settings	Boundary Type	Wall
	Location	TOP
Boundary Details	Wall Influence on Flow > Option	Free Slip

3. Click **OK**.

Wall Boundary

1. Create a new boundary condition named `WingSurface`.
2. Apply the following settings

Tab	Setting	Value
Basic Settings	Boundary Type	Wall
	Location	WING_Nodes*

*. Click the ellipsis  icon to select items if they do not appear in the drop-down list.


3. Click **OK**.

Creating Domain Interfaces

The imported mesh contains three regions which 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*
	Interface Side 2 > Region List	Primitive 2D, Primitive 2D B

*. Click the ellipsis  icon to select items if they do not appear in the drop-down list.

3. Click **OK**.

Setting Initial Values

For high speed compressible flow, the ANSYS CFX-Solver usually requires sensible initial conditions to be set for the 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	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]
	Initial Conditions > Turbulence Eddy Dissipation	(Selected)


3. Click **OK**.

Setting Solver Control

The residence time for the fluid is approximately:

$$70 \text{ [m]} / 600 \text{ [m s}^{-1}\text{]} = 0.117 \text{ [s]}$$

In the next step, you will start with a conservative time scale that gradually increases towards the fluid residence time as the residuals decrease. A user specified maximum time scale can be combined with an auto timescale in ANSYS CFX-Pre.


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

3. Click **OK**.

Writing the Solver (.def) File

Since this tutorial uses domain interfaces and the **Summarize Interface Data** toggle was selected, an information window is displayed that informs you of the connection type used for each domain interface.

1. Click *Write Solver File* .
2. Apply the following settings

Setting	Value
File name	WingSPS.def
Summarize Interface Data	(Selected)
Quit CFX-Pre *	(Selected)

*. If using ANSYS CFX-Pre in Standalone Mode.

3. Ensure **Start Solver Manager** is selected and click **Save**.
4. The **Interface Summary** dialog box is displayed. This displays information related to the summary of interface connections. Click **OK**.
5. If using Standalone Mode, quit ANSYS CFX-Pre, saving the simulation (**.cfx**) file at your discretion.

Obtaining a Solution using ANSYS CFX-Solver Manager

When ANSYS CFX-Pre has shut down, and the ANSYS CFX-Solver Manager has started, obtain a solution to the CFD problem by following the instructions below.

1. In the ANSYS CFX-Solver Manager, click **Start Run**.
2. Click **Yes** to post-process the results when the completion message appears at the end of the run.
3. If using Standalone Mode, quit ANSYS CFX-Solver Manager.

Viewing the Results in ANSYS CFX-Post

The following topics will be discussed:

- [Displaying Mach Information](#) (p. 162)
- [Displaying Pressure Information](#) (p. 163)
- [Displaying Temperature Information](#) (p. 163)
- [Displaying Pressure With User Vectors](#) (p. 163)

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. Select **View Towards -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
	Variable	Mach Number
	Range	User Specified
	Min	1
	Max	2
	# of Contours	21

- Click **Apply**.
- Clear the check box next to `SymP2Mach`.

Displaying Pressure Information

You will now create a contour plot that shows the pressure field.

- Create a new contour named `SymP2Pressure`.
- Apply the following settings

Tab	Setting	Value
Geometry	Locations	SymP2
	Variable	Pressure
	Range	Global

- Click **Apply**.
- Clear the check box next to `SymP2Pressure`.

Displaying Temperature Information

You can confirm that a significant energy loss occurs around the wing leading edge by plotting temperature on `SymP2`. The temperature at the wing tip is approximately 180 K higher than the inlet temperature.

- Create a new contour named `SymP2Temperature`.
- Apply the following settings

Tab	Setting	Value
Geometry	Locations	SymP2
	Variable	Temperature
	Range	Global

- Click **Apply**.
- Clear the check box next to `SymP2Temperature`.

Displaying Pressure With User Vectors

You can also try creating 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	$(\text{Pressure} + 101325[\text{Pa}]) * \text{Normal X}$
	Y Expression	$(\text{Pressure} + 101325[\text{Pa}]) * \text{Normal Y}$
	Z Expression	$(\text{Pressure} + 101325[\text{Pa}]) * \text{Normal Z}$

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.

Tutorial 9: Flow Through a Butterfly Valve

Introduction

This tutorial includes:

- [Tutorial 9 Features](#) (p. 165)
- [Overview of the Problem to Solve](#) (p. 166)
- [Defining a Simulation in ANSYS CFX-Pre](#) (p. 167)
- [Obtaining a Solution using ANSYS CFX-Solver Manager](#) (p. 180)
- [Viewing the Results in ANSYS CFX-Post](#) (p. 180)

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

- [Setting the Working Directory](#) (p. 1)
- [Changing the Display Colors](#) (p. 2)

Unless you plan on running a session file, you should copy the sample files used in this tutorial from the installation folder for your software (<CFXROOT>/examples/) to your working directory. This prevents you from overwriting source files provided with your installation. If you plan to use a session file, please refer to [Playing a Session File](#) (p. 167).

Sample files referenced by this tutorial include:

- **PipeValve.pre**
- **PipeValve_inlet.F**
- **PipeValveMesh.gtm**
- **PipeValveUserF.pre**

Tutorial 9 Features

This tutorial addresses the following features of ANSYS CFX.

Component	Feature	Details
ANSYS CFX-Pre	User Mode	General Mode
	Simulation 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)	
User Fortran		
Timestep	Auto Time Scale	
ANSYS CFX-Solver Manager	Power-Syntax	
ANSYS CFX-Post	Plots	Animation
		Default Locators
		Particle Track
		Point
		Slice Plane
	Other	Changing the Color Range
		MPEG Generation
		Particle Track Animation
		Quantitative Calculation
		Symmetry

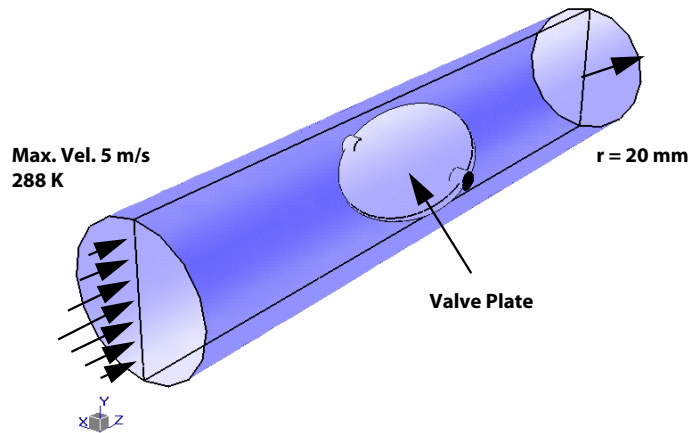
In this tutorial you will learn about:

- using a rough wall boundary condition in ANSYS CFX-Pre to simulate the pipe wall
- creating a fully developed inlet velocity profile using either the CFX Expression Language or a User CEL Function
- setting up a Particle Tracking simulation in ANSYS CFX-Pre to trace sand particles
- animating particle tracks in ANSYS CFX-Post to trace sand particles through the domain
- quantitative calculation of average static pressure in ANSYS CFX-Post on the outlet boundary

Overview of the Problem to Solve

In industry, 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 requirement and cost.



Flows can also contain particulates that affect the flow and cause erosion to pipe and valve components. The particle tracking capability of ANSYS CFX can be used to simulate these effects.

In this example, water flows through a 20 mm radius pipe with a rough internal surface. The equivalent sand grain roughness is 0.2 mm. The flow is controlled by a butterfly valve, which is set at an angle of 55° to the vertical axis. The velocity profile is assumed to be fully developed at the pipe inlet. The flow contains sand particles ranging in size from 50 to 500 microns.

Defining a Simulation in ANSYS CFX-Pre

The following sections describe the simulation setup in ANSYS CFX-Pre.

Playing a Session File

If you wish to skip past these instructions, and have ANSYS CFX-Pre set up the simulation automatically, you can select **Session > Play Tutorial** from the menu in ANSYS CFX-Pre, then run one of the following session files available for this tutorial:

- **PipeValve.pre** sets the inlet velocity profile using a CEL (ANSYS CFX Expression Language) expression.
- **PipeValveUserF.pre** sets the inlet velocity profile using a User CEL Function that is defined by a Fortran subroutine. This session file requires that you have the required Fortran compiler installed and set in your system path. 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.

If you choose to run a session file do so using the procedure described in earlier tutorials under [Playing the Session File and Starting ANSYS CFX-Solver Manager](#) (p. 87), and then proceed to [Obtaining a Solution using ANSYS CFX-Solver Manager](#) (p. 180) once the simulation setup is complete.

Creating a New Simulation

1. Start ANSYS CFX-Pre.
2. Select **File > New Simulation**.
3. Select **General** and click **OK**.
4. Select **File > Save Simulation As**.
5. Under **File name**, type `PipeValve`.
6. Click **Save**.

Importing the Mesh

1. Right-click `Mesh` and select **Import Mesh**.
2. Apply the following settings

Setting	Value
File name	PipeValveMesh.gtm

3. Click **Open**.

Defining the Properties of 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 property that needs to be defined is the density of the sand.

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 optimise CPU usage, you can create two sets of identical particles. The first set will be fully coupled and between 100 and 1000 particles will be used. This allows the particles to influence the flow field. The second set will use one-way coupling but a much higher number of particles will be used. This provides a more accurate calculation of the particle volume fraction and local forces on walls.

1. Click **Material**  then create a new material named `Sand Fully Coupled`.

- 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]
	Thermodynamic Properties > Specific Heat Capacity	(Selected)
	Thermodynamic Properties > Specific Heat Capacity > Specific Heat Capacity	0 [J kg ⁻¹ K ⁻¹]*
	Thermodynamic Properties > Reference State	(Selected)
	Thermodynamic Properties > Reference State > Option	Specified Point
	Thermodynamic Properties > Reference State > Ref. Temperature	300 [K]

*. This value is not used because heat transfer is not modeled in this tutorial.

- Click **OK**.
- Under **Materials**, right-click **Sand Fully Coupled** and select **Duplicate** from the shortcut menu.
- Name the duplicate **Sand One Way Coupled**.
- Click **OK**.

Sand One Way Coupled is created with properties identical to **Sand Fully Coupled**.

Creating the Domain

- Right click **Simulation** in the **Outline** tree view and ensure that **Automatic Default Domain** is selected. A domain named **Default Domain** should now appear under the **Simulation** branch.
- Double click **Default Domain** and apply the following settings

Tab	Setting	Value
General Options	Basic Settings > Fluids List	Water
	Basic Settings > Particle Tracking	(Selected)
	Basic Settings > Particle Tracking > Particles List	Sand Fully Coupled, Sand One Way Coupled
	Domain Models > Pressure > Reference Pressure	1 [atm]
Fluid Models	Heat Transfer > Option	None
	Turbulence > Option	k-Epsilon*

Tab	Setting	Value
Fluid Details	Sand Fully Coupled	(Selected)
	Sand Fully Coupled > Morphology > Option	Solid Particles
	Sand Fully Coupled > Morphology > Particle Diameter Distribution	(Selected)
	Sand Fully Coupled > Morphology > Particle Diameter Distribution > Option	Normal in Diameter by Mass
	Sand Fully Coupled > Morphology > Particle Diameter Distribution > Minimum Diameter	50e-6 [m]
	Sand Fully Coupled > Morphology > Particle Diameter Distribution > Maximum Diameter	500e-6 [m]
	Sand Fully Coupled > Morphology > Particle Diameter Distribution > Mean Diameter	250e-6 [m]
	Sand Fully Coupled > Morphology > Particle Diameter Distribution > Std. Deviation	70e-6 [m]
	Sand Fully Coupled > Erosion Model	(Selected)
	Sand Fully Coupled > Erosion Model > Option	Finnie
	Sand Fully Coupled > Erosion Model > Vel. Power Factor	2.0
	Sand Fully Coupled > Erosion Model > Reference Velocity	1 [m s ⁻¹]

*. The turbulence model only applies to the continuous phase and not the particle phases.

3. Apply the following settings

Tab	Setting	Value
Fluid Details	Sand One Way Coupled	(Selected)
	Sand One Way Coupled > Morphology > Option	Solid Particles
	Sand One Way Coupled > Morphology > Particle Diameter Distribution	(Selected)
	Sand One Way Coupled > Morphology > Particle Diameter Distribution > Option	Normal in Diameter by Mass
	Sand One Way Coupled > Morphology > Particle Diameter Distribution > Minimum Diameter	50e-6 [m]
	Sand One Way Coupled > Morphology > Particle Diameter Distribution > Maximum Diameter	500e-6 [m]
	Sand One Way Coupled > Morphology > Particle Diameter Distribution > Mean Diameter	250e-6 [m]
	Sand One Way Coupled > Morphology > Particle Diameter Distribution > Std. Deviation	70e-6 [m]
	Sand One Way Coupled > Erosion Model	(Selected)
	Sand One Way Coupled > Erosion Model > Option	Finnie
		Sand One Way Coupled > Erosion Model > Vel. Power Factor
	Sand One Way Coupled > Erosion Model > Reference Velocity	1 [m s ⁻¹]

4. Apply the following settings

Tab	Setting	Value
Fluid Details	Water	(Selected)
	Water > Morphology > Option	Continuous Fluid
Fluid Pairs	Fluid Pairs	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
	Fluid Pairs	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

5. Click **OK**.

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:

$$U = W_{\max} \left(1 - \frac{r}{R_{\max}} \right)^{\frac{1}{7}} \quad (\text{Eqn. 1})$$

where W_{\max} is the pipe centerline velocity, R_{\max} is the pipe radius, and r is the distance from the pipe centerline.

A non uniform (profile) boundary condition can be created by:

- Creating an expression using CEL that describes the inlet profile.
OR
- Creating a User CEL Function which uses a user subroutine (linked to the ANSYS CFX-Solver during execution) to describe the inlet profile.
OR
- Loading a BC profile file (a file which contains profile data).

Profiles created from data files are not used in this tutorial, but are used in the tutorial [Tutorial 3: Flow in a Process Injection Mixing Pipe](#) (p. 77).

In this tutorial, you use one of the first two methods listed above to define the velocity profile for the inlet boundary condition. The results from each method will be identical.

Using a CEL expression is the easiest way to create the profile. The User CEL Function method is more complex but is provided as an example of how to use this feature. For more complex profiles, it may be necessary to use a User CEL Function or a BC profile file.

To use the User CEL Function method, continue with this tutorial from [User CEL Function Method for the Inlet Velocity Profile](#) (p. 173). Note that you will need access to a Fortran compiler to be able to complete the tutorial by the User CEL Function method.

To use the expression method, continue with the tutorial from this point.

Expression Method for the Inlet Velocity Profile

1. Create the following expressions.

Name	Definition
Rmax	20 [mm]
Wmax	5 [m s ⁻¹]
Wprof	Wmax*(abs(1-r/Rmax) ^{0.143})

In the definition of w_{prof} , the variable r (radius) is a ANSYS CFX System Variable defined as:

$$r = \sqrt{x^2 + y^2} \quad (\text{Eqn. 2})$$

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.

You should now continue with the tutorial from [Creating the Boundary Conditions](#) (p. 175).

**User CEL
Function
Method for the
Inlet Velocity
Profile**

The Fortran subroutine has already been written for this tutorial.

Important: You must have the required Fortran compiler installed and set in your system path in order to run this part of the tutorial. If you do not have a Fortran compiler, you should use the expression method for defining the inlet velocity, as described in [Expression Method for the Inlet Velocity Profile](#) (p. 172). 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.

Compiling the Subroutine

1. Copy the subroutine `PipeValve_inlet.F` to your working directory. It is located in the `<CFXROOT>/examples/` directory.
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.

You can compile the subroutine and create the required library files used by the ANSYS CFX-Solver at any time before running the ANSYS 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 ANSYS CFX-Pre when creating a User CEL Function. The `cfx5mkext` command is used to create the required objects and libraries as described below.

3. From the main menu, select **Tools > Command Editor**.
4. Type the following in the **Command Editor** dialog box (make sure you do not miss the semi-colon at the end of the line):

```
! system ("cfx5mkext PipeValve_inlet.F") < 1 or die;
```

- This is equivalent to executing the following at an OS command prompt:
`cfx5mkext PipeValve_inlet.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 `< 1 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.

The output produced when this command is executed will be printed to your terminal window.

Note: You can use the `-double` option (that is, `cfx5mkext -double PipeValve_inlet.F`) to compile the subroutine for use with double precision.

A subdirectory will have been created in your working directory whose name is system dependent (for example, on IRIX it is named `irix`). 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.

6. Close the **Command Editor** dialog box.


Creating the Input Arguments

Next, you will create some values that will be used as input arguments when the subroutine is called.


1. Click *Expression* .
2. Set **Name** to `Wmax`, and then click **OK**.
3. Type `5 [m s^-1]` into the **Definition** box, and then click **Apply**.
The expression will be listed in the **Expressions** tree view.
4. Use the same method to create an expression named `Rmax` defined to be `20 [mm]`.


Creating the User CEL Function

Two steps are required to define a User CEL Function that uses the compiled Fortran subroutine. First, a User Routine that points to the Fortran subroutine will be created. Then a User CEL Function that points to the User Routine will be created.

1. From the main toolbar, click *User Routine* .
2. Set **Name** to `WprofRoutine`, and then click **OK**.
The **User Routine** details view appears.
3. Set **Option** to `User CEL Function`.
4. Set **Calling Name** to `inlet_velocity`.
 - 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.
5. Set **Library Name** to `PipeValve_inlet`.
 - 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.
6. Set **Library Path** to the directory where the `cfx5mkext` command was executed (usually the current working directory). For example:
 - UNIX: `/home/user/cfx/tutorials/PipeValve`.
 - Windows: `c:\user\cfx\tutorials\PipeValve`.

This can be accomplished quickly by clicking *Browse*  (next to **Library Path**), browsing to the appropriate folder in **Select Directory** (not necessary if selecting the working directory), and clicking **OK** (in **Select Directory**).

7. Click **OK** to complete the definition of the user routine.
8. Click *User Function* .
9. Set **Name** to `WprofFunction`, and then click **OK**.
The **Function** details view appears.

Important: You must not use the same name for the function and the routine.

10. Set **Option** to `User Function`.
11. Set **User Routine Name** to `WprofRoutine`.
12. Set **Argument Units** to `[m s^-1]`, `[m]`, `[m]`. These are the units for the three input arguments: `Wmax`, `r`, and `Rmax`.

Set **Result Units** to `[m s^-1]`, since the result will be a velocity for the inlet.

1. Click **OK** to complete the User Function specification.

You can now use the user function (`WprofFunction`) in place of a velocity value by entering the expression `WprofFunction(Wmax, r, Rmax)` (although it only makes sense for the `W` component of the inlet velocity in this tutorial).

In the definition of `WprofFunction`, the variable `r` (radius) is a system variable defined as:

$$r = \sqrt{x^2 + y^2} \tag{Eqn. 3}$$


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.

Creating the Boundary Conditions

- Inlet Boundary**
1. Create a new boundary condition named `inlet`.
 2. Apply the following settings

Tab	Setting	Value
Basic Settings	Boundary Type	Inlet
	Location	inlet

Tab	Setting	Value
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 -OR- WprofFunction(Wmax, r, Rmax)*
Fluid Values [†]	Boundary Conditions	Sand Fully Coupled
	Sand Fully Coupled > Particle Behavior > Define Particle Behavior	(Selected)
	Sand Fully Coupled > Mass and Momentum > Option	Cart. Vel. Components [‡]
	Sand Fully Coupled > Mass And Momentum > U	0 [m s ⁻¹]
	Sand Fully Coupled > Mass And Momentum > V	0 [m s ⁻¹]
	Sand Fully Coupled > Mass And Momentum > W	Wprof -OR- WprofFunction(Wmax, r, Rmax)**
	Sand Fully Coupled > Particle Position > Option	Uniform Injection
	Sand Fully Coupled > Particle Position > Number of Positions > Option	Direct Specification
	Sand Fully Coupled > Particle Position > Number of Positions > Number	200
	Sand Fully Coupled > Particle Mass Flow > Mass Flow Rate	0.01 [kg s ⁻¹]
Fluid Values	Boundary Conditions	Sand One Way Coupled
	Sand One Way Coupled > Particle Behavior > Define Particle Behavior	(Selected)
	Sand One Way Coupled > Mass and Momentum > Option	Cart. Vel. Components ^{††}
	Sand One Way Coupled > Mass And Momentum > U	0 [m s ⁻¹]
	Sand One Way Coupled > Mass And Momentum > V	0 [m s ⁻¹]
	Sand One Way Coupled > Mass And Momentum > W	Wprof -OR- WprofFunction(Wmax, r, Rmax) ^{‡‡}
	Sand One Way Coupled > Particle Position > Option	Uniform Injection
	Sand One Way Coupled > Particle Position > Number of Positions > Option	Direct Specification
	Sand One Way Coupled > Particle Position > Number of Positions > Number	5000
	Sand One Way Coupled > Particle Position > Particle Mass Flow Rate > Mass Flow Rate	0.01 [kg s ⁻¹]

- *. Use the **Expressions** details view  to enter either `wprof` if using the expression method, or `wprofFunction(wmax, r, Rmax)` if using the User CEL Function method.
- †. 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.
- ‡. Instead of manually specifying the same velocity profile as the fluid, you can also select the Zero Slip Velocity option.
- **.
- ††. Instead of manually specifying the same velocity profile as the fluid, you can also select the Zero Slip Velocity option.
- ‡‡. 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.

Outlet Boundary

1. Create a new boundary condition 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**.

Symmetry Plane Boundary

1. Create a new boundary condition named `symP`.
2. Apply the following settings

Tab	Setting	Value
Basic Settings	Boundary Type	Symmetry
	Location	symP

3. Click **OK**.

Pipe Wall Boundary

1. Create a new boundary condition named `pipe wall`.
2. Apply the following settings

Tab	Setting	Value
Basic Settings	Boundary Type	Wall
	Location	pipe wall

Tab	Setting	Value
Boundary Details	Wall Roughness > Option	Rough Wall
	Roughness Height	0.2 [mm]*
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
	Boundary Conditions > Sand Fully Coupled > Velocity > Parallel Coeff.	1
	Boundary Conditions	Sand One Way Coupled
	Boundary Conditions > Sand One Way Coupled > Velocity > Option	Restitution Coefficient
	Boundary Conditions > Sand One Way Coupled > Velocity > Perpendicular Coeff.	0.8
	Boundary Conditions > Sand One Way Coupled > Velocity > Parallel Coeff.	1

*. Make sure that you change the units to millimetres. The thickness of the first element should be of the same order as the roughness height.

3. Click **OK**.


Editing the Default Boundary Condition

1. In the **Outline** tree view, edit the boundary condition 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
	Boundary Conditions	Sand One Way Coupled
	Boundary Conditions > Sand One Way Coupled > Velocity > Perpendicular Coeff.	0.9

3. Click **OK**.

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 > 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 -OR- WprofFunction(Wmax, r, Rmax)*
	Initial Conditions > Turbulence Eddy Dissipation	(Selected)

*. Use Enter Expression  to enter w_{prof} if using the Expression method; enter $WprofFunction(Wmax, r, Rmax)$ if using the User CEL Function method.

3. Click **OK**.


Setting Solver Control

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

Tab	Setting	Value
Basic Settings	Advection Scheme > Option	Specified Blend Factor
	Advection Scheme > Blend Factor	0.75
Particle Control	Particle Integration > Maximum Tracking Time	(Selected)
	Particle Integration > Maximum Tracking Time > Value	10 [s]
	Particle Integration > Maximum Tracking Distance	(Selected)
	Particle Integration > Maximum Tracking Distance > Value	10 [m]
	Particle Integration > Max. Num. Integration Steps	(Selected)
	Particle Integration > Max. Num. Integration Steps > Value	10000
	Particle Integration > Max. Particle Intg. Time Step	(Selected)
	Particle Integration > Max. Particle Intg. Time Step > Value	1e+10 [s]

3. Click **OK**.

Writing the Solver (.def) File

1. Click *Write Solver File* .
2. Apply the following settings:

Setting	Value
File name	PipeValve.def
Quit CFX-Pre *	(Selected)

*. If using ANSYS CFX-Pre in Standalone Mode.

3. Ensure **Start Solver Manager** is selected and click **Save**.
4. If using Standalone Mode, quit ANSYS CFX-Pre, saving the simulation (.cfx) file at your discretion.

Obtaining a Solution using ANSYS CFX-Solver Manager

When ANSYS CFX-Pre has shut down and ANSYS CFX-Solver Manager has started, you can obtain a solution to the CFD problem by using the following procedure.

Note: If you followed the User CEL Function method, and you wish to run this tutorial in distributed parallel on machines with different architectures, you must first compile the `PipeValve_inlet.F` subroutine on all architectures.


1. Ensure the **Define Run** dialog box is displayed and click **Start Run**.
2. Click **Yes** to post-process the results when the completion message appears at the end of the run.
3. If using Standalone Mode, quit ANSYS CFX-Solver Manager.

Viewing the Results in ANSYS CFX-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.

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 selections

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] [†]

*. This is statistically better than Sand Fully Coupled.Erosion Rate Density since many more particles were calculated for Sand One Way Coupled.

†. 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.

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 10 tracks as lines from the inlet to outlet. Info shows information about the total number of tracks, index range and the track numbers which 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

3. Click **Apply**.

Erosion on the Pipe Wall

The User Specified range for coloring will be set to resolve areas of stress on the pipe wall near of the valve.

1. Clear the check box next to Res PT for Sand Fully Coupled.
2. Clear the check box next to Default Domain Default.
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**.

Particle Track Symbols


1. Clear 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	Draw Symbols	(Selected)
	Draw Symbols > Max Time	0 [s]
	Draw Symbols > Min Time	0 [s]
	Draw Symbols > Interval	0.07 [s]
	Draw Symbols > Symbol	Fish3D
	Draw Symbols > Symbol Size	0.5




4. Clear **Draw Tracks**.
5. Click **Apply**.
Symbols are placed at the start of each track.

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. Select **Tools > Animation** or click *Animation* .
2. Select **Quick Animation**.
3. Select *Res PT for Sand Fully Coupled*:
4. 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.

5. Select **Loop**.
6. Deselect *Repeat forever*  and ensure **Repeat** is set to 1.
7. Select **Save MPEG**.
8. Click *Browse*  and enter `tracks.mpg` as the file name.
9. Click *Play the animation* .
10. If prompted to overwrite an existing movie, click **Overwrite**.
The animation plays and builds an `.mpg` file.
11. Close the **Animation** dialog box.

Performing Quantitative Calculations

On the outlet boundary condition you created in ANSYS 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 ANSYS CFX-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.

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 condition.

Other Features The geometry was created using a symmetry plane. You 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.

1. When you have finished viewing the results, quit ANSYS CFX-Post.

Tutorial 10: Flow in a Catalytic Converter

Introduction

This tutorial includes:

- [Tutorial 10 Features](#) (p. 185)
- [Overview of the Problem to Solve](#) (p. 186)
- [Defining a Simulation in ANSYS CFX-Pre](#) (p. 187)
- [Obtaining a Solution using ANSYS CFX-Solver Manager](#) (p. 193)
- [Viewing the Results in ANSYS CFX-Post](#) (p. 194)

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

- [Setting the Working Directory](#) (p. 1)
- [Changing the Display Colors](#) (p. 2)

Unless you plan on running a session file, you should copy the sample files used in this tutorial from the installation folder for your software (<CFXROOT>/examples/) to your working directory. This prevents you from overwriting source files provided with your installation. If you plan to use a session file, please refer to [Playing a Session File](#) (p. 187).

Sample files referenced by this tutorial include:

- `CatConv.pre`
- `CatConvHousing.hex`
- `CatConvMesh.gtm`

Tutorial 10 Features

This tutorial addresses the following features of ANSYS CFX.

Component	Feature	Details
ANSYS CFX-Pre	User Mode	General Mode
	Simulation Type	Steady State
	Fluid Type	Ideal Gas
	Turbulence Model	k-Epsilon
	Heat Transfer	Isothermal
	Subdomains	Resistance Source
	Boundary Conditions	Inlet (Subsonic)
		Outlet (Subsonic)
		Wall: No-Slip
	Domain Interfaces	Fluid-Fluid (No Frame Change)
Timestep	Physical Time Scale	
ANSYS CFX-Post	Plots	Contour
		Default Locators
		Outline Plot (Wireframe)
		Polyline
		Slice Plane
		Vector
	Other	Chart Creation
		Data Export
		Title/Text
		Viewing the Mesh

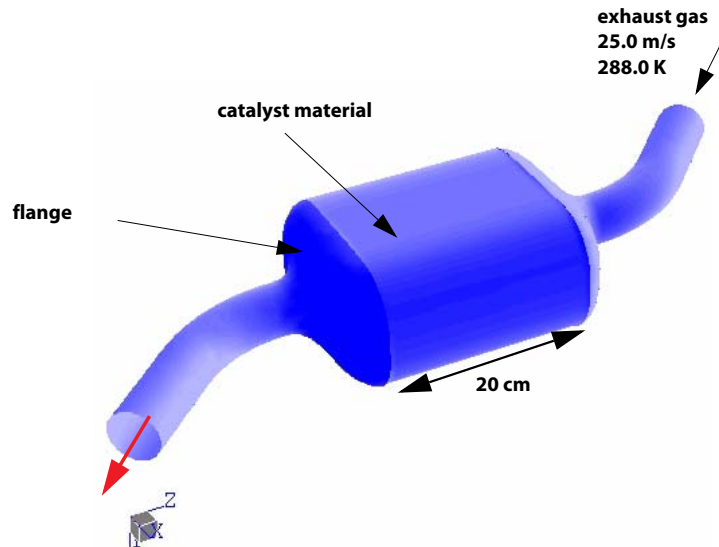
In this tutorial you will learn about:

- Using multiple meshes in ANSYS 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 through the domain in ANSYS CFX-Post.
- Exporting data from a line locator to a file.

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. The exhaust gases flow through the honeycomb structure and a pressure gradient is established between the inlet and outlet.



In this tutorial, a catalytic converter is modeled without chemical reactions in order to determine the pressure drop. The inlet flange (joining the pipe to the catalyst) is designed to distribute exhaust gas evenly across the catalyst material.

A hexahedral mesh for the housing, which was created in ICM-Hexa, is provided.

The different meshes are connected together in ANSYS CFX-Pre. You will import each mesh then create a domain, which spans all of them. Within the converter, a subdomain is added to model a honeycomb structure using a directional loss model. The physics is then specified in the same way as for other tutorials.

Defining a Simulation in ANSYS CFX-Pre

The following sections describe the simulation setup in ANSYS CFX-Pre.

Playing a Session File

If you wish to skip past these instructions, and have ANSYS CFX-Pre set up the simulation automatically, you can select **Session > Play Tutorial** from the menu in ANSYS CFX-Pre, then run the session file: `CatConv.pre`. After you have played the session file as described in earlier tutorials under [Playing the Session File and Starting ANSYS CFX-Solver Manager](#) (p. 87), proceed to [Obtaining a Solution using ANSYS CFX-Solver Manager](#) (p. 193).

Creating a New Simulation

1. Start ANSYS CFX-Pre.
2. Select **File > New Simulation**.

3. Select **General** and click **OK**.
4. Select **File > Save Simulation As**.
5. Under **File name**, type `CatConv`.
6. Click **Save**.

Importing the Meshes

The catalytic converter is comprised of three distinct parts:

- The inlet section (pipe and flange).
- The outlet section (pipe and flange).
- The catalyst (or monolith).

Next you will import a generic inlet/outlet section and the catalyst housing from provided files.

Housing Section The first mesh that you will import is the hexahedral mesh for the catalyst housing, created in ICEM-Hexa, named `CatConvHousing.hex`. This mesh was created using units of centimetres; however, the units are not stored with the mesh file for this type of mesh. You must set the mesh import units to cm when importing the mesh into ANSYS CFX-Pre so that the mesh remains the intended size. The imported mesh has a width in the x-direction of 21 cm and a length in the z-direction of 20 cm.

1. Right-click `Mesh` and select **Import Mesh**.
2. Apply the following settings

Setting	Value
File type	All Types
Definition > Mesh Format	ICEM CFD
File name	CatConvHousing.hex
Definition > Mesh Units	cm

3. Click **Open**.

Pipe and Flange Section This mesh was created in units of centimetres. When importing GTM files, ANSYS CFX-Pre uses the units used in the mesh file.

1. Right-click `Mesh` and select **Import Mesh** to import the second section.
2. Apply the following settings

Setting	Value
File type	CFX Mesh (gtm)
File name	CatConvMesh.gtm

3. Click **Open**.

You only need to import this mesh once, as you will be copying and rotating the flange through 180 degrees in the next step to create the inlet side pipe and flange.

Applying a Transform

The pipe and flange are located at the outlet end of the housing. The flange will be rotated about an axis that points in the y-direction and is located at the center of the housing.

1. Right-click `CatConvMesh.gtm` and select **Transform Mesh**. The **Mesh Transformation Editor** dialog box appears.
2. Apply the following settings

Tab	Setting	Value
Definition	Apply Rotation > Rotation Option	Rotation Axis
	Apply Rotation > From	0, 0, 0.16
	Apply Rotation > To	0, 1, 0.16*
	Apply Rotation > Rotation Angle	180 [degree]
	Multiple Copies	(Selected)
	Multiple Copies > # of Copies	1

*. This specifies an axis located at the center of the housing parallel to the y-axis.

3. Click **OK**.

Creating a Union Region

Three separate regions now exist, but since there is no relative motion between each region, you only need to create a single domain. This can be done by simply using all three regions in the domain **Location** list or, as in this case, by using the **Region** details view to create a union of the three regions.


1. Create a new composite region named `CatConverter`.
2. Apply the following settings

Tab	Setting	Value
Basic Settings	Dimension (Filter)	3D
	Region List	B1.P3, B1.P3 2, LIVE

3. Click **OK**.

Creating the Domain

For this simulation you will use an isothermal heat transfer model and assume turbulent flow.

1. Click *Domain*  and set the name to `CatConv`.
2. Apply the following settings

Tab	Setting	Value
General Options	Basic Settings > Location	CatConverter
	Basic Settings > Fluid List	Air Ideal Gas
	Domain Models > Pressure > Reference Pressure	1 [atm]
Fluid Models	Heat Transfer > Option	Isothermal
	Heat Transfer > Fluid Temperature	600 [K]

3. Click **OK**.

Creating a Subdomain to Model the Catalyst Structure


The catalyst-coated honeycomb structure will be modeled using a subdomain with a directional source of resistance.

For quadratic resistances, the pressure drop is modeled using:

$$\frac{\partial p}{\partial x_i} = -K_Q |U| U_i \quad (\text{Eqn. 1})$$

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. Select **Insert > Subdomain** from the main menu or click *Subdomain* .
2. In the **Insert Subdomain** dialog box, type `catalyst`.
3. Apply the following settings

Tab	Setting	Value
Basic Settings	Location	LIVE*
Sources†	Sources	(Selected)
	Sources > Momentum Source/Porous Loss	(Selected)
	Sources > Momentum Source/Porous Loss > Directional Loss	(Selected)

*. This is the entire housing section.

†. Used to set sources of momentum, resistance and mass for the subdomain (Other sources are available for different problem physics).

4. Apply the following settings in the **Directional Loss** section

Setting	Value
Streamwise Direction > X Component	0
Streamwise Direction > Y Component	0
Streamwise Direction > Z Component	1

Setting	Value
Streamwise Loss > Option	Linear and Quadratic Coefs
Streamwise Loss > Quadratic Resistance Coefficient	(Selected)
Streamwise Loss > Quadratic Resistance Coefficient > Quadratic Coefficient	650 [kg m ⁻⁴]

- Click **OK**.

Creating Boundary Conditions

- Inlet Boundary**
- Create a new boundary condition named `Inlet`.
 - Apply the following settings

Tab	Setting	Value
Basic Settings	Boundary Type	Inlet
	Location	PipeEnd 2
Boundary Details	Mass and Momentum > Normal Speed	25 [m s ⁻¹]

- Click **OK**.

- Outlet Boundary**
- Create a new boundary condition named `Outlet`.
 - Apply the following settings

Tab	Setting	Value
Basic Settings	Boundary Type	Outlet
	Location	PipeEnd
Boundary Details	Mass and Momentum > Option	Static Pressure
	Mass and Momentum > Relative Pressure	0 [Pa]

- Click **OK**.
The remaining surfaces are automatically grouped into the default no slip wall boundary condition.

Creating the Domain Interfaces

Domain interfaces are used to define the connecting boundaries between meshes where the faces do not match or when a frame change occurs. Meshes are 'glued' together using the General Grid Interface (GGI) functionality of ANSYS CFX. Different types of GGI connections can be made. In this case, you require a simple Fluid-Fluid Static connection (no Frame Change). Other options allow you to change reference frame across the interface or create a periodic boundary with dissimilar meshes on each periodic face.

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.

Inlet Pipe / Housing Interface

1. Create a new domain interface named `InletSide`.
2. Apply the following settings

Tab	Setting	Value
Basic Settings	Interface Side 1 > Region List	FlangeEnd 2
	Interface Side 2 > Region List	INLET

3. Click **OK**.

Outlet Pipe / Housing Interface


1. Create a new domain interface named `OutletSide`.
2. Apply the following settings

Tab	Setting	Value
Basic Settings	Interface Side 1 > Region List	FlangeEnd
	Interface Side 2 > Region List	OUTLET

3. Click **OK**.

Setting Initial Values

A sensible guess for the initial velocity is to set it to the expected velocity through the catalyst housing. As the inlet velocity is 25 [m s⁻¹] and the cross sectional area of the inlet and housing are known, you can apply conservation of mass to obtain an approximate velocity of 2 [m s⁻¹] through the housing.

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	-2 [m s ⁻¹]
	Initial Conditions > Turbulence Eddy Dissipation	(Selected)

3. Click **OK**.

Setting Solver Control

Assuming velocities of 25 [m s⁻¹] in the inlet and outlet pipes, and 2 [m s⁻¹] in the catalyst housing, an approximate fluid residence time of 0.1 [s] can be calculated. A sensible timestep of 0.04 [s] (1/4 to 1/2 of the fluid residence time) will be applied.

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.


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**.

Writing the Solver (.def) File

While writing the solver file, you will use the **Summarize Interface Data** option to display information about the connection type used for each domain interface.

1. Click *Write Solver File* .
2. Apply the following settings

Setting	Value
File name	CatConv.def
Summarize Interface Data	(Selected)
Quit CFX-Pre *	(Selected)

*. If using ANSYS CFX-Pre in Standalone Mode.

3. Ensure **Start Solver Manager** is selected and click **Save**.
4. Once ANSYS CFX-Solver Manager launches, return to ANSYS CFX-Pre.
The **Interface Summary** dialog box is displayed. This displays information related to the summary of interface connections.
5. Click **OK**.
6. If using Standalone Mode, quit ANSYS CFX-Pre, saving the simulation (.cfx) file at your discretion.

Obtaining a Solution using ANSYS CFX-Solver Manager

When ANSYS CFX-Pre has shut down and the ANSYS CFX-Solver Manager has started, you can obtain a solution to the CFD problem by following the instructions below:

1. Ensure **Define Run** is displayed.
2. Click **Start Run**.
ANSYS 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. Click **Yes** to post-process the results.
4. If using Standalone Mode, quit ANSYS CFX-Solver Manager.

Viewing the Results in ANSYS CFX-Post

When ANSYS CFX-Post opens, you will need to experiment with the **Edge Angle** setting for the `Wireframe` object in order to view an appropriate amount of the mesh.

Under the **Outline** tab, several interface boundaries are available. The two connections between the catalyst housing mesh and the mesh for the inlet and outlet pipes have two interface boundaries each, one for each side of the connection.

1. Zoom in so the geometry fills the viewer.
2. In the **Outline** tree view, edit `InletSide Side 1`.
3. Apply the following settings

Tab	Setting	Value
Render	Draw Faces	(Cleared)
	Draw Lines	(Selected)
	Draw Lines > Color Mode	User Specified
	Draw Lines > Line Color	(Red)

4. Click **Apply**.
5. In the **Outline** tree view, edit `InletSide Side 2`.
6. Apply the following settings

Tab	Setting	Value
Render	Draw Faces	(Cleared)
	Draw Lines	(Selected)
	Draw Lines > Color Mode	User Specified
	Draw Lines > Line Color	(Green)

7. Click **Apply**.
8. In the **Outline** tree view, clear `Wireframe` to hide it.
9. Right-click a blank area in the viewer and select **Predefined Camera > View Towards -Z**.

You should now have a clear view of the tetrahedral / prism and hexahedral mesh on each side of the interface. The General Grid Interface (GGI) capability of ANSYS CFX was used to produce a connection between these two 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). The extent of each side of the interface does not have to match to allow a GGI connection to be made.

Creating User Locations and Plots

1. In the **Outline** tree view, select `Wireframe` to show it.
2. In the **Outline** tree view, clear both `InletSide Side 1` and `InletSide Side 2`.

Creating a Slice Plane

1. Create a new plane named `Plane 1`.
2. Apply the following settings

Tab	Setting	Value
Geometry	Definition > Method	ZX Plane
Color	Mode	Variable
	Variable	Pressure

3. Click **Apply**.
4. Right-click a blank area in the viewer and select **Predefined Camera > View Towards -Y**.

Creating a Contour Plot

The pressure falls steadily throughout the main body of the catalytic converter. You can confirm this with a contour plot.

1. Clear `Plane 1` in the **Outline** tab.
2. Create a new contour plot named `Contour 1`.
3. Apply the following settings

Tab	Setting	Value
Geometry	Locations	Plane 1
	Variable	Pressure
	# of Contours	30
Render	Draw Faces	(Cleared)

4. Click **Apply**.

Creating a Vector Plot Using the Slice Plane

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 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.

Suppose for now that you want to see if the pressure drop is linear by plotting a line graph of pressure against the z-coordinate. In this case you will use ANSYS CFX-Post to produce the graph, but you could also export the data, then read it into any standard plotting package.

Graphs are produced using the chart object, but before you can create the chart you must define the points at which you require the data. To define a set of points in a line, you can use the polyline object.

Creating 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 (e.g., 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. In the **Outline** tree view, clear `Contour 1` and `Vector 1`.
2. Create a new polyline named `Polyline 1`.
3. Apply the following settings

Tab	Setting	Value
Geometry	Method	Boundary Intersection
	Boundary List	CatConv Default, Inlet, Outlet *
	Intersect With	Plane 1
Color	Color	(Yellow)
Render	Line Width	3

*. Click the ellipsis icon to select multiple items using the <Ctrl> key.

4. Click **Apply**.
5. Right-click a blank area in the viewer and select **Predefined Camera > Isometric View (Y up)**.

Creating a Chart

Now that a polyline has been defined, a chart can be created. Charts are defined by creating chart line objects. A chart line is listed in the tree view beneath the chart object to which it belongs.

1. Create a new chart named `Chart 1`.
2. Apply the following settings

Tab	Setting	Value
Chart	Title	Pressure Drop through a Catalytic Converter
Chart Line 1	Line Name	Pressure Drop
	Location	Polyline 1
	X Axis > Variable	Z
	Y Axis > Variable	Pressure
Appearance	Appearance > Symbols	Rectangle
Appearance	Sizes > Line	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 $Z=0.05$ to $Z=0.25$. 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 $Z=0.25$).

4. If required, in the **Outline** tree view, select **Contour 1**, **Polyline 1**, and **Vector 1**.
5. Click the **3D Viewer** tab, then right-click a blank area and select **Predefined Camera > View Towards +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.

6. Under **Report**, expand **Chart 1**, and edit **Chart Line 1**.
7. Apply the following settings

Tab	Setting	Value
Chart Line 1	X Axis > Variable	Chart Count*

*. This is the data point number (e.g. 1,2,3,4...), it does NOT represent the distance between each point along the polyline.

8. Click **Apply**.

Exporting Data

1. From the main menu, select **File > Export**.
2. Apply the following settings

Tab	Setting	Value
Options	Locations	Polyline 1
	Export Geometry Information	(Selected)*
	Select Variables	Pressure
Formatting	Precision	3

*. 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. This file can be opened in any text editor. You can use the exported data file to plot charts in other software.

4. When finished, quit ANSYS CFX-Post.

Tutorial 11: Non-Newtonian Fluid Flow in an Annulus

Introduction

This tutorial includes:

- [Tutorial 11 Features](#) (p. 200)
- [Overview of the Problem to Solve](#) (p. 201)
- [Defining a Simulation in ANSYS CFX-Pre](#) (p. 201)
- [Obtaining a Solution using ANSYS CFX-Solver Manager](#) (p. 205)
- [Viewing the Results in ANSYS CFX-Post](#) (p. 206)

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

- [Setting the Working Directory](#) (p. 1)
- [Changing the Display Colors](#) (p. 2)

Unless you plan on running a session file, you should copy the sample files used in this tutorial from the installation folder for your software (<CFXROOT>/examples/) to your working directory. This prevents you from overwriting source files provided with your installation. If you plan to use a session file, please refer to [Playing a Session File](#) (p. 201).

Sample files referenced by this tutorial include:

- **NonNewton.pre**
- **NonNewtonMesh.gtm**

Tutorial 11 Features

This tutorial addresses the following features of ANSYS CFX.

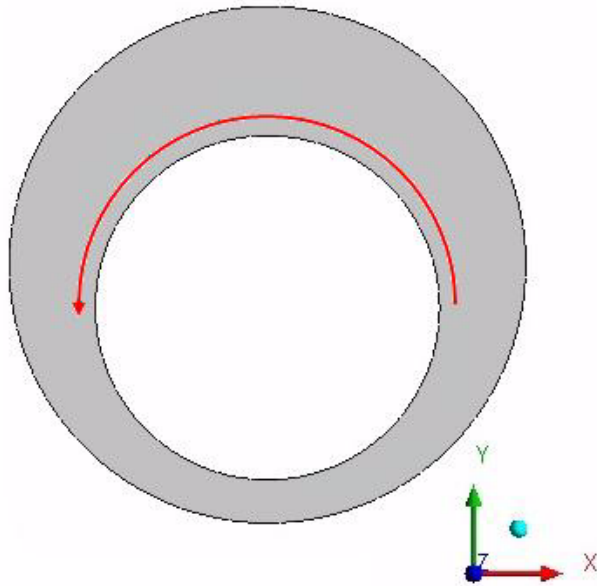
Component	Feature	Details	
ANSYS CFX-Pre	User Mode	General Mode	
	Simulation Type	Steady State	
	Fluid Type	General Fluid	
	Domain Type	Single Domain	
	Turbulence Model	Laminar	
	Heat Transfer	None	
	Boundary Conditions		Symmetry Plane
			Wall: No-Slip
			Wall: Moving
		CEL (CFX Expression Language)	
	Timestep	Auto Time Scale	
ANSYS CFX-Post	Plots	Sampling Plane	
		Slice Plane	
		Vector	

In this tutorial you will learn about:

- Using CFX Expression Language (CEL) to define the properties of a shear-thickening fluid.
- Using the Moving Wall feature to apply a rotation to the fluid at a wall boundary.

Overview of the Problem to Solve

In this example a non-Newtonian, shear-thickening liquid rotates in a 2D eccentric annular pipe gap. The motion, shown by the arrow, is brought about solely by viscous fluid interactions caused by the rotation of the inner pipe.



Defining a Simulation in ANSYS CFX-Pre

The following sections describe the simulation setup in ANSYS CFX-Pre.

Playing a Session File

If you wish to skip past these instructions, and have ANSYS CFX-Pre set up the simulation automatically, you can select **Session > Play Tutorial** from the menu in ANSYS CFX-Pre, then run the session file: **NonNewtonon.pre**. After you have played the session file as described in earlier tutorials under [Playing the Session File and Starting ANSYS CFX-Solver Manager](#) (p. 87), proceed to [Obtaining a Solution using ANSYS CFX-Solver Manager](#) (p. 205).

Creating a New Simulation

1. Start ANSYS CFX-Pre.
2. Select **File > New Simulation**.
3. Select **General** and click **OK**.
4. Select **File > Save Simulation As**.
5. Under **File name**, type `NonNewtonon`.
6. Click **Save**.

Importing the Mesh

1. Right-click **Mesh** and select **Import Mesh**.
2. Apply the following settings

Setting	Value
File name	NonNewtonMesh.gtm

3. Click **Open**.

Creating an Expression for Shear Rate Dependent Viscosity

You can use an expression to define the dependency of fluid properties on other variables. In this case, the fluid does not obey the simple linear Newtonian relationship between shear stress and shear strain rate. The general relationship for the fluid you will model is given by:

$$\mu = K\gamma^{n-1} \quad (\text{Eqn. 1})$$

where γ is the shear strain rate and K and n are constants. For your fluid, $n=1.5$ and this results in shear-thickening behavior of the fluid, i.e., the viscosity increases with increasing shear strain rate. The shear strain rate is available as a ANSYS CFX-Pre System Variable (`sstrnr`).

In order to describe this relationship using CEL, the dimensions must be consistent on both sides of the equation. Clearly this means that K must have dimensions and requires units to satisfy the equation. If the units of viscosity are $\text{kg m}^{-1} \text{s}^{-1}$, and those of γ are s^{-1} , then the expression is consistent if the units of K are $\text{kg m}^{-1} \text{s}^{-0.5}$.

1. Create the following expressions, remembering to click **Apply** after each is defined.

Name	Definition
K	10.0 [kg m ⁻¹ s ^{-0.5}]
n	1.5

You should bound the viscosity to ensure that it remains physically meaningful. To do so, you will create two additional parameters that will be used to guarantee the value of the shear strain rate.

2. Create the following expressions for upper and lower bounds.

Name	Definition
UpperS	100 [s ⁻¹]
LowerS	1.0E-3 [s ⁻¹]
ViscEqn	$K * (\min(\text{UpperS}, \max(\text{sstrnr}, \text{LowerS}))^{(n-1)})$

3. Close the **Expressions** tab.

Creating a New Fluid


1. Create a new material named `myfluid`.
2. Apply the following settings

Tab	Setting	Value
Basic Settings	Thermodynamic State	(Selected)
Material Properties	Equation of State > Molar Mass	1 [kg kmol ⁻¹] [*]
	Equation of State > Density	1.0E+4 [kg m ⁻³]
	Specific Heat Capacity	(Selected)
	Specific Heat Capacity > Specific Heat Capacity	0 [J kg ⁻¹ K ⁻¹] [†]
	Reference State	(Selected)
	Reference State > Option	Specified Point
	Reference State > Ref. Temperature	25 [C]
	Reference State > Reference Pressure	1 [atm]
	Transport Properties > Dynamic Viscosity	(Selected)
	Transport Properties > Dynamic Viscosity > Dynamic Viscosity	ViscEqn

- *. This is not the correct Molar Mass value, but this material property will not be used by the ANSYS CFX-Solver for this case. In other cases it will be used.
- †. This is not the correct value for specific heat, but this property will not be used in the ANSYS CFX-Solver.

3. Click **OK**.

Creating the Domain

1. Click *Domain*  and set the name to `NonNewton`.
2. Apply the following settings to `NonNewton`

Tab	Setting	Value
General Options	Basic Settings > Fluids List	myfluid
	Domain Models > Pressure > Reference Pressure	1 [atm]
Fluid Models	Heat Transfer > Option	Isothermal
	Heat Transfer > Fluid Temperature	25 C
	Turbulence > Option	None (Laminar)

3. Click **OK**.

Creating the Boundary Conditions

Wall Boundary

1. Create a new boundary condition named `rotwall`.
2. Apply the following settings

Tab	Setting	Value
Basic Settings	Boundary Type	Wall
	Location	rotwall
Boundary Details	Wall Influence on Flow > Wall Velocity	(Selected)
	Wall Influence on Flow > Wall Velocity > Option	Rotating Wall
	Wall Influence on Flow > Wall Velocity > Angular Velocity	31.33 [rev min ⁻¹]

3. Click **OK**.

Symmetry Plane Boundary

1. Create a new boundary condition 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 condition 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 condition.

Setting Initial Values


A reasonable initial guess for the velocity field is a value of zero throughout the domain.

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**.

Setting Solver Control


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

Tab	Setting	Value
Basic Settings	Advection Scheme > Option	Specific Blend Factor
	Advection Scheme > Blend Factor	1*
	Convergence Control > Max. Iterations	50
	Convergence Criteria > Residual Target	1e-05

*. This is the most accurate but least robust advection scheme.

3. Click **OK**.

Writing the Solver (.def) File

1. Click *Write Solver File* .
2. Apply the following settings:

Setting	Value
File name	NonNewton.def
Quit CFX-Pre*	(Selected)

*. If using ANSYS CFX-Pre in Standalone Mode.

3. Ensure **Start Solver Manager** is selected and click **Save**.
4. If using Standalone Mode, quit ANSYS CFX-Pre, saving the simulation (.cfx) file at your discretion.

Obtaining a Solution using ANSYS CFX-Solver Manager

When ANSYS CFX-Pre has shut down and the ANSYS CFX-Solver Manager has started, you can obtain a solution to the CFD problem by following the instructions below:

1. Ensure **Define Run** is displayed.
2. Click **Start Run**.
ANSYS 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. Click **Yes** to post-process the results.
4. If using Standalone Mode, quit ANSYS CFX-Solver Manager.

Viewing the Results in ANSYS CFX-Post

In this tutorial you have used CEL to create an expression for the dynamic viscosity. If you now perform calculations or color graphics objects using the Dynamic Viscosity variable, its values will have been calculated from the expression you defined in ANSYS CFX-Pre.

These steps instruct the user on how to create a vector plot to show the velocity values in the domain.

1. Right-click a blank area in the viewer and select **Predefined Camera > View Towards -Z** from the shortcut menu.
2. Create a new plane named `Plane 1`.
3. Apply the following settings

Tab	Setting	Value
Geometry	Definition > Method	Point and Normal
	Definition > Point	0, 0, 0.02
	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	Draw Faces	(Cleared)

4. Click **Apply**.
5. Create a new vector plot named `Vector 1`.
6. Apply the following settings

Tab	Setting	Value
Geometry	Definition > Locations	Plane 1
	Definition > Variable	Velocity
Symbol	Symbol Size	3

7. Click **Apply**.
8. Try creating some plots of your own, including one that shows the variation of dynamic viscosity.
9. When you have finished, quit ANSYS CFX-Post.

Tutorial 12:

Flow in an Axial Rotor/Stator

Introduction

This tutorial includes:

- [Tutorial 12 Features](#) (p. 208)
- [Overview of the Problem to Solve](#) (p. 209)
- [Defining a Frozen Rotor Simulation in ANSYS CFX-Pre](#) (p. 210)
- [Obtaining a Solution to the Frozen Rotor Model](#) (p. 214)
- [Viewing the Frozen Rotor Results in ANSYS CFX-Post](#) (p. 215)
- [Setting up a Transient Rotor-Stator Calculation](#) (p. 216)
- [Obtaining a Solution to the Transient Rotor-Stator Model](#) (p. 219)
- [Viewing the Transient Rotor-Stator Results in ANSYS CFX-Post](#) (p. 220)

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

- [Setting the Working Directory](#) (p. 1)
- [Changing the Display Colors](#) (p. 2)

Unless you plan on running a session file, you should copy the sample files used in this tutorial from the installation folder for your software (`<CFXROOT>/examples/`) to your working directory. This prevents you from overwriting source files provided with your installation. If you plan to use a session file, please refer to [Playing a Session File](#) (p. 211).

Sample files referenced by this tutorial include:

- **Axial.pre**
- **AxialIni.pre**
- **AxialIni_001.res**
- **rotor.grd**
- **stator.gtm**

Tutorial 12 Features

This tutorial addresses the following features of ANSYS CFX.

Component	Feature	Details
ANSYS CFX-Pre	User Mode	Turbo Wizard
	Simulation 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
		Periodic
Transient Rotor Stator		
Timestep	Physical Time Scale	
	Transient Example	
Transient Results File		
ANSYS CFX-Solver Manager	Restart	
	Parallel Processing	
ANSYS CFX-Post	Plots	Animation
		Isosurface
		Surface Group
	Turbo Post	
	Other	Changing the Color Range
		Chart Creation
		Instancing Transformation
		MPEG Generation
		Quantitative Calculation
		Time Step Selection
Transient Animation		

In this tutorial you will learn about:

- Using the Turbo Wizard in ANSYS 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.

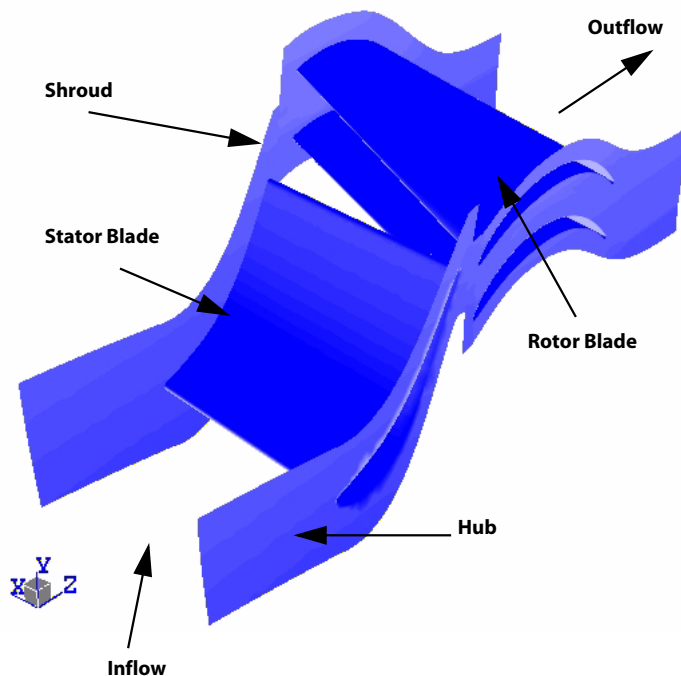
- Creating a transient animation showing domain movement in ANSYS CFX-Post.

Overview of the Problem to Solve

The following tutorial demonstrates the versatility of GGI and MFR in ANSYS CFX-Pre by combining two dissimilar meshes. The first mesh to be imported (the rotor) was created in CFX-TASCflow. This is combined with a second mesh (the stator) which was created using ANSYS CFX-Mesh.

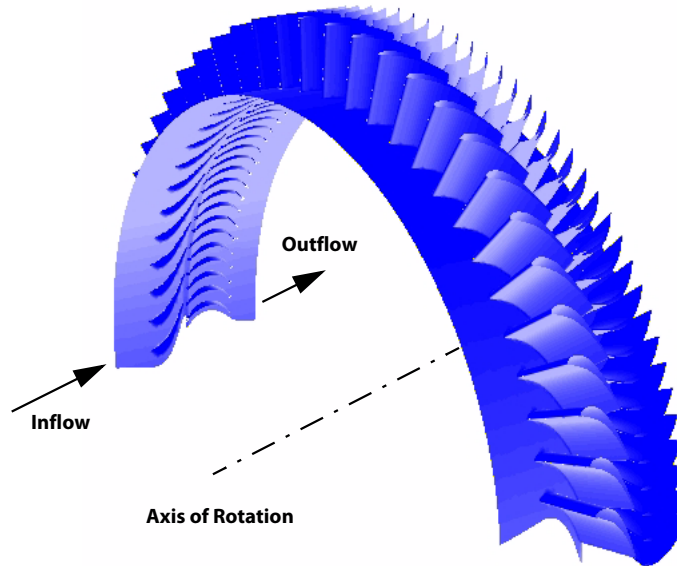
The geometry to be modeled consists of a single stator blade passage and two rotor blade passages. The rotor rotates about the Z-axis while the stator is stationary. Periodic boundaries are used to allow only a small section of the full geometry to be modeled.

Figure 1 Geometry subsection



At the change in reference frame between the rotor and stator, two different interface models are considered. First a solution is obtained using a Frozen Rotor model. After viewing the results from this simulation, the latter is modified to use a transient rotor-stator interface model. The Frozen Rotor solution is used as an initial guess for the transient rotor-stator simulation.

The full geometry contains 60 stator blades and 113 rotor blades. To help you visualize how the modeled geometry fits into the full geometry, the following figure shows approximately half of the full geometry. The Inflow and Outflow labels show the location of the modeled section in .



As previously indicated, the modeled geometry contains two rotor blades and one stator blade. 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 ($360^\circ/60$ blades), while in the rotor blade passage a 6.372° section is being modeled ($2 \times 360^\circ/113$ 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.

The periodic boundary conditions will introduce an additional approximation since they cannot be periodic when a pitch change occurs.

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.

Defining a Frozen Rotor Simulation in ANSYS CFX-Pre

The following sections describe the simulation setup in ANSYS CFX-Pre.

Playing a Session File

If you wish to skip past these instructions, and have ANSYS CFX-Pre set up the simulation automatically, you can select **Session > Play Tutorial** from the menu in ANSYS CFX-Pre, then run the session file: `AxialIni.pre`. After you have played the session file as described in earlier tutorials under [Playing the Session File and Starting ANSYS CFX-Solver Manager](#) (p. 87), proceed to [Obtaining a Solution to the Frozen Rotor Model](#) (p. 214).

Creating a New Simulation

This tutorial will use the Turbomachinery wizard in ANSYS CFX-Pre. This pre-processing mode is designed to simplify the setup of turbomachinery simulations.

1. Start ANSYS CFX-Pre.
2. Select **File > New Simulation**.
3. Select **Turbomachinery** and click **OK**.
4. Select **File > Save Simulation As**.
5. Under **File name**, type `AxialIni`.
6. Click **Save**.

Basic Settings

1. Set **Machine Type** to `Axial Turbine`.
2. Click **Next**.

Component Definition

Two new components are required. As they are created, meshes are imported.

1. Right-click in the blank area and select **New 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*

*. You may have to select the `CFX Mesh` option under **File 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 > File	rotor.grd*

*. You may have to select the CFX-TASCflow option under **File 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 condition 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.

- Click **Next**.

Physics Definition

In this section, you will set properties of the fluid domain and some solver parameters.

- Apply the following settings

Tab	Setting	Value
Physics Definition	Fluid	Air Ideal Gas
	Simulation Type > Type	Steady State
	Model Data > Reference Pressure	0.25 [atm]
	Model Data > Heat Transfer	Total Energy
	Model Data > Turbulence	k-Epsilon
	Boundary Templates > P-Total Inlet Mass Flow Outlet	(Selected)
	Boundary Templates > P-Total	0 [atm]
	Boundary Templates > T-Total	340 [K]
	Boundary Templates > Mass Flow Rate	0.06 [kg s ⁻¹]
	Interface > Default Type	Frozen Rotor
	Solver Parameters > Convergence Control	Physical Timescale
	Solver Parameters > Physical Timescale	0.002 [s]*

*. This time scale is approximately equal to $1 / \omega$, which is often appropriate for rotating machinery applications.

- Click **Next**.

Interface Definition

ANSYS 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**.

Boundary Definition


ANSYS 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**.

Final Operations

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

Writing the Solver (.def) File

1. Click *Write Solver File* .
2. Apply the following settings:

Setting	Value
File name	AxialIni.def
Quit CFX-Pre *	(Selected)

*. If using ANSYS CFX-Pre in Standalone Mode.

3. Ensure **Start Solver Manager** is selected and click **Save**.
4. If using Standalone Mode, quit ANSYS CFX-Pre, saving the simulation (`.cfx`) file at your discretion.

You should see ANSYS CFX-Solver Manager appear.

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 ANSYS CFX to run in parallel is available. For details, see [Tutorial 5: Flow Around a Blunt Body](#) (p. 109).

You can solve this example using Serial, Local Parallel or Distributed Parallel.

- [Obtaining a Solution in Serial](#) (p. 214)
- [Obtaining a Solution With Local Parallel](#) (p. 214)
- [Obtaining a Solution with Distributed Parallel](#) (p. 215)

Obtaining a Solution in Serial


If you do not have a license to run ANSYS CFX in parallel you can run in serial by clicking the **Start Run** button when ANSYS 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.
ANSYS 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.
2. Click **Yes** to start ANSYS CFX-Post.
3. If using Standalone Mode, quit ANSYS CFX-Solver Manager.

When you are finished, proceed to [Viewing the Frozen Rotor Results in ANSYS CFX-Post](#) (p. 215).


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 `PVM Local Parallel` in the **Define Run** dialog box.
This is the recommended method for most applications.
2. If required, click *Add Partition*  to add more partitions.
By default, 2 partitions are assigned.
3. Click **Start Run**.
4. Click **Yes** to post-process the results when the completion message appears at the end of the run.
5. If using Standalone Mode, quit ANSYS CFX-Solver Manager.

When you are finished, proceed to [Viewing the Frozen Rotor Results in ANSYS CFX-Post](#) (p. 215).

Obtaining a Solution with Distributed Parallel

1. Set **Run Mode** to `PVM Distributed Parallel` in the **Define Run** dialog box. 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. Click **Yes** to post-process the results when the completion message appears at the end of the run.
7. If using Standalone Mode, quit ANSYS CFX-Solver Manager.

Viewing the Frozen Rotor Results in ANSYS CFX-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.

Initializing Turbo-Post

To initialize Turbo-Post, the properties of each component must be entered. This includes entering information about the inlet, outlet, hub, shroud, blade and periodic regions.

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**.

The **Turbo** tree view shows the two components in domains `R1` and `S1`. In this case, the initialization works without problems. If there was a problem initializing a component, this would be indicated in the tree view.

Viewing Three Domain Passages

Next, you will create an instancing transformation to plot three blade passages for the stator and six blade passages for the rotor.

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**.

Blade Loading Turbo Chart

In this section, you will create a plot of pressure around the stator blade at a given spanwise location.

1. 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 ANSYS CFX-Post.

Setting up a Transient Rotor-Stator Calculation

This section describes the step-by-step definition of the flow physics in ANSYS CFX-Pre. The existing 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 Simulation in ANSYS CFX-Pre](#) (p. 210) before proceeding with the transient rotor-stator simulation.

Playing a Session File

If you wish to skip past these instructions, and have ANSYS CFX-Pre set up the simulation automatically, you can select **Session > Play Tutorial** from the menu in ANSYS CFX-Pre, then run the session file: **Axial1.pre**. After you have played the session file as described in earlier tutorials under [Playing the Session File and Starting ANSYS CFX-Solver Manager](#) (p. 87), proceed to [Obtaining a Solution to the Transient Rotor-Stator Model](#) (p. 219).

Note: The session file creates a new simulation named **Axial1.cfx** and will not modify the existing database. It also copies the required initial values files from the examples directory to the current working directory.

Opening the Existing Simulation

This step involves opening the original simulation and saving it to a different location.

1. Start ANSYS CFX-Pre.
2. Open the results file named **Axial1Ini_001.res**.
3. Save the simulation as **Axial1.cfx** in your working directory.
4. Select **Tools > Turbo Mode**.
Basic Settings is displayed

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 timesteps. This is generally too few timesteps to obtain high quality results, but is sufficient for tutorial purposes. The timestep size is calculated as follows:

```
Rotational Speed = 523.6 rad/s
Rotor Pitch Modelled = 2*(2π/113) = 0.1112 rad
Time to pass through 1 pitch = 0.1112/523.6 = 2.124e-4 s
```

Since 10 time steps are used over this interval each timestep should be 2.124e-5 s.

1. Click **Next**.
Component Definition is displayed.
2. Click **Next**.
Physics Definition is displayed.
3. Apply the following settings

Tab	Setting	Value
Physics Definition	Fluid	Air Ideal Gas
	Simulation Type > Type	Transient
	Simulation Type > Total Time	2.124e-4 [s]*
	Simulation Type > Time Steps	2.124e-5 [s]†
	Interface > Default Type	Transient Rotor Stator


*. This gives 10 timesteps of 2.124e-5 s

†. 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.

4. Click **Next**.
Interface Definition is displayed.
5. Click **Next**.
Boundary Definition is displayed.
6. Click **Next**.
Final Operations is displayed.
7. Ensure that **Operation** is set to `Enter General Mode`.
8. Click **Finish**.
Initial values are required, but will be supplied later using a results file.

Setting Output Control

1. Click *Output Control* .
2. Click the **Trn Results** tab.
3. Create a new transient result with the name `Transient Results 1`.
4. Apply the following settings to `Transient Results 1`

Setting	Value
Option	Selected Variables
Output Variables List*	Pressure, Velocity, Velocity in Stn Frame
Output Frequency > Option	Time Interval
Output Frequency > Time Interval	2.124e-5 [s]

*. Use the <Ctrl> key to select more than one variable.

5. Click **OK**.

Writing the Solver (.def) File

1. Click *Write Solver File* .

A warning will appear, due to a lack of initial values.

2. Click **Yes**.
Initial values are required, but will be supplied later using a results file.
3. Apply the following settings:

Setting	Value
File name	Axial.def
Quit CFX-Pre*	(Selected)

*. If using ANSYS CFX-Pre in Standalone Mode.


4. Ensure **Start Solver Manager** is selected and click **Save**.
5. If using Standalone Mode, quit ANSYS CFX-Pre, saving the simulation (.cfx) file at your discretion.

Obtaining a Solution to the Transient Rotor-Stator Model

When the ANSYS CFX-Solver Manager has started you will need to specify an initial values file before starting the ANSYS CFX-Solver.

Serial Solution

If you do not have a license, or do not want to run ANSYS CFX in parallel, you can run it in serial. Solution time in serial is similar to the first part of this tutorial.

1. Under **Initial Values File**, click *Browse* .
2. Select **AxialIni_001.res**.
3. Click **Open**.
4. Click **Start Run**.
5. You may see a notice that the mesh from the initial values file will be used. This mesh is the same as in the definition file. Click **OK** to continue.
ANSYS 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.
6. Click **Yes** to post-process the results when the completion message appears at the end of the run.
7. If using Standalone Mode, quit ANSYS CFX-Solver Manager.
When you are finished, continue with [Monitoring the Run](#) (p. 220).

Parallel Solution

You can solve this example using either local parallel or distributed parallel, in the same way as in the first part of this tutorial. For details, see [Obtaining a Solution to the Frozen Rotor Model](#) (p. 214).

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.

Viewing the Transient Rotor-Stator Results in ANSYS CFX-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.

Initializing Turbo-Post

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.

Displaying a Surface of Constant Span

1. 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.

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**.

Creating a Turbo Surface Midway Between the Hub and Shroud

1. Create a **Turbo Surface** from the Insert drop down menu with a **Constant Span** and value of 0.5.

2. Under the **Color** panel select **Variable** and set it to **Pressure** with a user specified range of -10000 [Pa] to -7000 [Pa].


Setting up Instancing Transformations

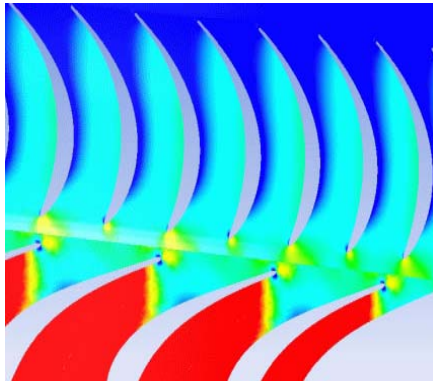
Next, you will use instancing transformations to view a larger section of the model. The properties for each domain have already been entered during the initialization phase, so only the number of instances needs to be set.



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.






Creating a Transient Animation

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. Clear **Visibility** for *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, click *New*  to create `KeyFrameNo1`.
8. Highlight `KeyFrameNo1`, then set **# of Frames** to 9.
9. Use the **Timestep Selector** to load the final timestep.

10. In the **Animation** dialog box, click **New**  to create `KeyframeNo2`.
11. Click **More Animation Options**  to expand the **Animation** dialog box.
12. 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.
13. In the expanded **Animation** dialog box, select **Save MPEG**.
14. Click **Browse** , next to the **Save MPEG** box and then set the file name to an appropriate file name.
15. 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 ANSYS CFX-Post to finish loading the objects for this frame before proceeding.
16. Click **Play the animation** .
 - It takes a while for the animation to complete.
 - To view the MPEG 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.

Further Postprocessing

You can use the Turbo Calculator to produce a report on the performance of the turbine.

1. Edit the `Gas Turbine Performance` macro in the **Turbo** tree view.
2. Set **Ref Radius** to `0.4575` and leave other settings at their default values.
3. Click **Calculate**.
4. Click **View Report**.

Tutorial 13: Reacting Flow in a Mixing Tube

Introduction

This tutorial includes:

- [Tutorial 13 Features](#) (p. 223)
- [Overview of the Problem to Solve](#) (p. 224)
- [Outline of the Process](#) (p. 224)
- [Defining a Simulation in ANSYS CFX-Pre](#) (p. 225)
- [Obtaining a Solution using ANSYS CFX-Solver Manager](#) (p. 237)
- [Viewing the Results in ANSYS CFX-Post](#) (p. 237)

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

- [Setting the Working Directory](#) (p. 1)
- [Changing the Display Colors](#) (p. 2)

Unless you plan on running a session file, you should copy the sample files used in this tutorial from the installation folder for your software (<CFXROOT>/examples/) to your working directory. This prevents you from overwriting source files provided with your installation. If you plan to use a session file, please refer to [Playing a Session File](#) (p. 225).

Sample files referenced by this tutorial include:

- **Reactor.pre**
- **ReactorExpressions.ccl**
- **ReactorMesh.gtm**

Tutorial 13 Features

This tutorial addresses the following features of ANSYS CFX.

Component	Feature	Details	
ANSYS CFX-Pre	User Mode	General Mode	
	Simulation 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		
ANSYS CFX-Post	Plots	Isosurface	
		Slice Plane	
	Other	Changing the Color Range	

In this tutorial you will learn about:

- Creating and using a multicomponent fluid in ANSYS CFX-Pre.
- Using CEL to model a reaction in ANSYS CFX-Pre.
- Using an algebraic additional variable to model a scalar distribution.
- Using a subdomain as the basis for component sources.

Overview of the Problem to Solve

Reaction engineering is one of the main 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 in modeling basic reacting flows using a multicomponent fluid model.

Outline of the Process

The model is a mixing tube into which acid and alkali are injected through side holes. The reaction to be modeled is:



The tube is modeled as an axisymmetric section.

The reaction between acid and alkali is represented as a single step irreversible liquid-phase reaction



Reagent A (dilute sulphuric 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.

The composition and pH of the mixture within the tube are principal quantities of interest to be predicted by the model.

The flow is assumed to be fully turbulent and turbulence is assumed to have a significant effect on the process. The process is also exothermic.

Defining a Simulation in ANSYS CFX-Pre

The following sections describe the simulation setup in ANSYS CFX-Pre.

Playing a Session File

If you wish to skip past these instructions, and have ANSYS CFX-Pre set up the simulation automatically, you can select **Session > Play Tutorial** from the menu in ANSYS CFX-Pre, then run the session file: **Reactor.pre**. After you have played the session file as described in earlier tutorials under [Playing the Session File and Starting ANSYS CFX-Solver Manager](#) (p. 87), proceed to [Obtaining a Solution using ANSYS CFX-Solver Manager](#) (p. 237).

Creating a New Simulation

1. Start ANSYS CFX-Pre.
2. Select **File > New Simulation**.
3. Select **General** and click **OK**.
4. Select **File > Save Simulation As**.
5. Under **File name**, type `Reactor`.
6. Click **Save**.

Importing the Mesh

1. Right-click `Mesh` and select **Import Mesh**.
2. Apply the following settings

Setting	Value
File name	ReactorMesh.gtm

3. Click **Open**.

Creating a Multicomponent Fluid

In addition to providing template fluids, ANSYS CFX allows you to create custom fluids for use in all your ANSYS CFX models. These fluids 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, ANSYS 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; this material is already defined since it is commonly used. 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.

Acid properties

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

Tab	Setting	Value
Material Properties	Thermodynamic Properties > Equation of State > Molar Mass	19.52 [kg kmol ⁻¹]*
	Thermodynamic Properties > Equation of State > Density	1080 [kg m ⁻³]
	Thermodynamic Properties > Specific Heat Capacity	(Selected)
	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 > Thermal Conductivity	0.6 [W m ⁻¹ K ⁻¹]

*. The Molar Masses for the three materials created are only set for completeness since they are not used when solving this problem.

3. Click **OK**.

Alkali properties

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	Thermodynamic Properties > Equation of State > Molar Mass	20.42 [kg kmol ⁻¹]
	Thermodynamic Properties > Equation of State > Density	1130 [kg m ⁻³]
	Thermodynamic Properties > Specific Heat Capacity	(Selected)
	Thermodynamic Properties > Specific Heat Capacity > Specific Heat Capacity	4190 [J kg ⁻¹ K ⁻¹]
	Transport Properties > Dynamic Viscosity	(Selected)
	Transport Properties > Dynamic Viscosity > Dynamic Viscosity	0.001 [kg m ⁻¹ s ⁻¹]
	Transport Properties > Thermal Conductivity	(Selected)
	Transport Properties > Thermal Conductivity > Thermal Conductivity	0.6 [W m ⁻¹ K ⁻¹]

3. Click **OK**.

Product of the reaction properties

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	Thermodynamic Properties > Equation of State > Molar Mass	21.51 [kg kmol ⁻¹]
	Thermodynamic Properties > Equation of State > Density	1190 [kg m ⁻³]
	Thermodynamic Properties > Specific Heat Capacity	(Selected)
	Thermodynamic Properties > Specific Heat Capacity > Specific Heat Capacity	4190 [J kg ⁻¹ K ⁻¹]
	Transport Properties > Dynamic Viscosity	(Selected)
	Transport Properties > Dynamic Viscosity > Dynamic Viscosity	0.001 [kg m ⁻¹ s ⁻¹]
	Transport Properties > Thermal Conductivity	(Selected)
	Transport Properties > Thermal Conductivity > Thermal Conductivity	0.6 [W m ⁻¹ K ⁻¹]

3. Click **OK**.

Fluid properties

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**.

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	Units	[kg kg ⁻¹]

- Click **OK**.

This additional variable is now available for use when you create or modify a domain.

Defining the Reaction

Reactions and reaction kinetics can be modeled using CFX Expression Language (CEL), together with appropriate settings for Component sources. This section shows you how to develop an Eddy Break Up (EBU) type term using CEL to simulate the reaction between acid and alkali.

Reaction Source Terms

The reaction and reaction rate are modeled using a basic Eddy Break Up formulation for the component and energy sources, so that, for example, the transport equation for mass fraction of acid is

$$\begin{aligned}
 & \frac{\partial}{\partial t}(\rho m f_{\text{acid}}) + \nabla \cdot (\rho U m f_{\text{acid}}) - \nabla \cdot (\rho D_A \nabla m f_{\text{acid}}) \\
 & = -4\rho \frac{\varepsilon}{k} \min\left(m f_{\text{acid}}, \frac{m f_{\text{alkali}}}{i}\right)
 \end{aligned}
 \tag{Eqn. 3}$$

where $m f$ is mass fraction, D_A is the kinematic diffusivity (set above) and i is the stoichiometric ratio. The right hand side represents the source term applied to the transport equation for the mass fraction of acid. The left hand side consists of the transient, advection and diffusion terms.

For acid-alkali reactions, the stoichiometric ratio is usually based on volume fractions. To correctly model the reaction using an Eddy Break Up formulation based on mass fractions, you must calculate the stoichiometric ratio based on mass fractions.

In this tutorial the reaction is modeled by introducing source terms for the acid, alkali and product components. You can now also model this type of flow more easily using a reacting mixture as your fluid. There is also a tutorial example using a reacting mixture. For details, see [Tutorial 18: Combustion and Radiation in a Can Combustor](#) (p. 299).

Technical Note (Reference Only)

In ANSYS CFX, Release 11.0, 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 timestep 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

$$C = \frac{\partial S}{\partial \phi} \quad (\text{Eqn. 4})$$

Where this derivative cannot be computed easily,

$$C = \frac{S}{\phi} \quad (\text{Eqn. 5})$$

may be sufficient to ensure convergence.

Another useful recipe for C is

$$C = -\frac{\rho}{\tau} \quad (\text{Eqn. 6})$$

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 ($\partial S / \partial \phi > 0$) or sources that do not depend directly on the solved variable ϕ .

Calculating pH

The pH (or acidity) of the mixture is a function of the mass fraction of acid, alkali and product. For the purposes of this calculation, acid is assumed to be dilute and fully dissociated into its respective ions (H^+ and X^-); alkali is assumed to be dilute and fully dissociated into its respective ions (Y^+ and OH^-); product is assumed to be a salt solution including further H^+ and OH^- ions in a stoichiometric ratio.

The concentrations of hydrogen and hydroxyl ions can be calculated from the mass fractions of the components using the following expressions:

$$[H^+]_{\text{acid}} = \alpha \rho \left(mf_{\text{acid}} + \frac{mf_{\text{prod}}}{1+i} \right) = [X^-] \quad (\text{Eqn. 7})$$

$$[OH^-]_{\text{alkali}} = \beta \rho \left(mf_{\text{alkali}} + \frac{imf_{\text{prod}}}{1+i} \right) = [Y^+] \quad (\text{Eqn. 8})$$

where α and β are the X^- ion and Y^+ ion concentrations in the acid and alkali respectively. For this problem, α is set to $1.0E-05$ kmole X^- per kg of acid, and $\beta = \alpha/i$. Applying charge conservation and equilibrium conditions,

$$[H^+] + [Y^+] = [X^-] + [OH^-] \quad (\text{Eqn. 9})$$

$$[H^+][OH^-] = K_W \quad (\text{Eqn. 10})$$

gives the following quadratic equation for free hydrogen ion concentration:

$$[H^+]([H^+] + [Y^+] - [X^-]) = K_W \quad (\text{Eqn. 11})$$

$$[H^+]^2 + ([Y^+] - [X^-])[H^+] - K_W = 0 \quad (\text{Eqn. 12})$$

$$pH = -\log_{10}[H^+] \quad (\text{Eqn. 13})$$

where K_W is the equilibrium constant ($1.0 \times 10E-14$ kmoles² m⁻⁶).

The quadratic equation can be solved for $[H^+]$ using the equation

$$[H^+] = \frac{-b + \sqrt{b^2 - 4ac}}{2a} \text{ where } a = 1, b = [Y^+] - [X^-] \text{ and } c = -K_W.$$

Creating expressions to model the reaction

You can create the expressions required to model the reaction sources and pH by either reading them in from a file or by defining them in the **Expressions** workspace. Note that the expressions used here do not refer to a particular fluid since there is only a single fluid. In a multiphase simulation you must prefix variables with a fluid name, for example **Mixture.acid.mf** instead of **acid.mf**.

In this tutorial the expressions can be imported from a file to avoid typing them.

Reading expressions from a file

1. Select **File > Import CCL**.
2. Ensure that **Import Method** is set to **Append**.
3. Select **ReactorExpressions.cc1**, which should be in your working directory.
4. Click **Open**.

Note that the expressions have been loaded.

Creating the Domain

1. Right click **Simulation** in the **Outline** tree view and ensure that **Automatic Default Domain** is selected. 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
General Options	Basic Settings > Domain Type	Fluid Domain
	Basic Settings > Fluids List	mixture
	Domain Models > Pressure > Reference Pressure	1 [atm]
Fluid Models	Heat Transfer > Option	Thermal Energy
	Component Details	acid
	Component Details > acid > Option	Transport Equation
	Component Details > acid > Kinematic Diffusivity	(Selected)
	Component Details > acid > Kinematic Diffusivity > Kinematic Diffusivity	0.001 [m ² s ⁻¹]

- Use the same **Option** and **Kinematic Diffusivity** settings for `alkali` and `product` as you have just set for `acid`.
- For `Water`, set **Option** to `Constraint` as follows

Tab	Setting	Value
Fluid Models	Component Details	Water
	Component Details > 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.

- Apply the following settings

Tab	Setting	Value
Fluid Models	Additional Variable Details > MixturePH	(Selected)
	Additional Variable Details > MixturePH > Option	Algebraic Equation
	Additional Variable Details > MixturePH > Value	pH

- Click **OK**.

Creating a Subdomain to Model the Chemical Reactions

To provide the correct modeling for the chemical reaction you need to define 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.

- Create a new subdomain named `sources`.
- Apply the following settings

Tab	Setting	Value
Sources	Sources	(Selected)
	Sources > Equation Sources	acid.mf
	Sources > Equation Sources > acid.mf	(Selected)
	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 > Source	AlkaliSource
	Sources > Equation Sources > alkali.mf > Source Coefficient	(Selected)
	Sources > Equation Sources > alkali.mf > Source Coefficient > Source Coefficient	AlkaliSourceCoeff
	Sources > Equation Sources	Energy
	Sources > Equation Sources > Energy	(Selected)
	Sources > Equation Sources > Energy > Source	HeatSource
	Sources > Equation Sources	product.mf
	Sources > Equation Sources > product.mf	(Selected)
	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]	

3. Click **OK**.

Creating the Boundary Conditions

Water Inlet Boundary

1. Create a new boundary condition named `InWater`.
2. Apply the following settings

Tab	Setting	Value
Basic Settings	Boundary Type	Inlet
	Location	InWater
Boundary Details	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**.

Acid Inlet Boundary

1. Create a new boundary condition named `InAcid`.

- Apply the following settings

Tab	Setting	Value
Basic Settings	Boundary Type	Inlet
	Location	InAcid
Boundary Details	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	

- Click **OK**.

Alkali Inlet Boundary

The inlet area for the alkali is twice that of the acid and it also enters at a higher velocity. The result is an acid-to-alkali volume inflow ratio of 1:2.667. Recall that a stoichiometric ratio of 2.7905 was specified based on mass fractions. When the density of the acid (1080 [kg m³]) and alkali (1130 [kg m³]) are considered, the acid-to-alkali mass flow ratio can be calculated as 1:2.7905. You are therefore providing enough acid and alkali to produce a neutral solution if they react together completely.

- Create a new boundary condition named `InAlkali`.
- Apply the following settings

Tab	Setting	Value
Basic Settings	Boundary Type	Inlet
	Location	InAlkali
Boundary Details	Mass and Momentum > Normal Speed	2.667 [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	

- Click **OK**.

Outlet Boundary

- Create a new boundary condition named `out`.
- 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**.

Symmetry Boundary

1. Create a new boundary condition 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 condition named `sym2`.
5. Apply the following settings

Tab	Setting	Value
Basic Settings	Boundary Type	Symmetry
	Location	sym2

6. Click **OK**.
The default adiabatic wall boundary condition will automatically be applied to the remaining unspecified boundary.

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.

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 > Turbulence Eddy Dissipation	(Selected)
	Initial Conditions > Turbulence Eddy Dissipation > Option	Automatic
	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
	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**.

Setting Solver Control

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


Tab	Setting	Value
Basic Settings	Advection Scheme > Option	Specific Blend Factor
	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	0.01 [s]*

*. The length of mixing tube is 0.06 [m] and inlet velocity is 2 [m s⁻¹]. An estimate of the dynamic time scale is 0.03 [s]. An appropriate timestep would be 1/4 to 1/2 of this value.

3. Click **OK**.

Note: At this point, you might see a physics validation message regarding a change in the advection scheme. This change will not affect the outcome of the simulation; you will still be able to run this simulation in the ANSYS CFX-Solver.

Writing the Solver (.def) File

1. Click *Write Solver File* .
2. Apply the following settings:

Setting	Value
File name	Reactor.def
Quit CFX-Pre *	(Selected)

*. If using ANSYS CFX-Pre in Standalone Mode.

3. Ensure **Start Solver Manager** is selected and click **Save**.
4. If using Standalone Mode, quit ANSYS CFX-Pre, saving the simulation (.cfx) file at your discretion.

Obtaining a Solution using ANSYS CFX-Solver Manager

When the ANSYS CFX-Solver Manager has started, obtain a solution to the CFD problem by following the instructions below.

Using the double precision ANSYS CFX-Solver executable is recommended for this case:

1. Ensure **Define Run** is displayed.
2. Select **Show Advanced Controls**. On the **Solver** tab, select **Double Precision** under **Executable Settings**.
3. Click **Start Run**.
ANSYS 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 ANSYS CFX-Solver Manager.

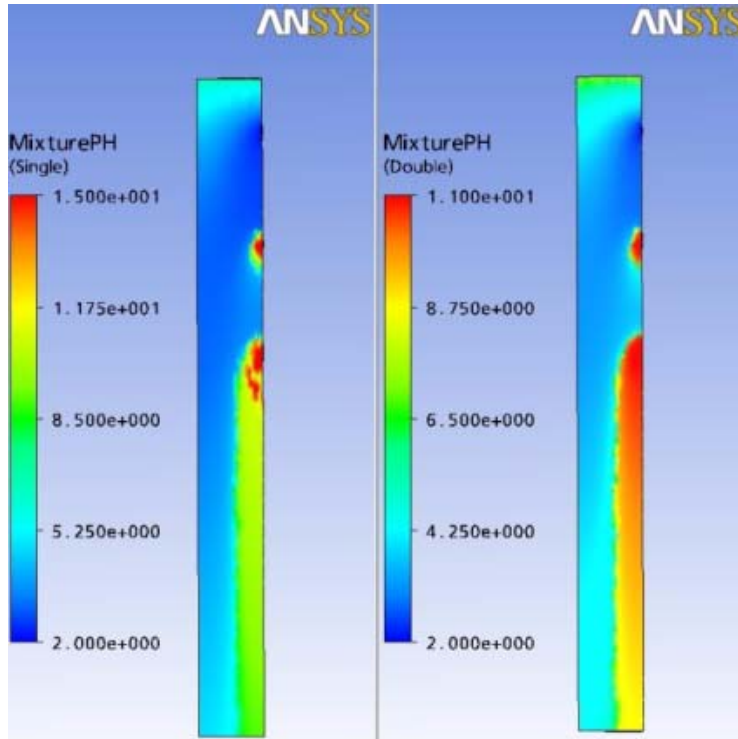
Viewing the Results in ANSYS CFX-Post

Try the following:

- Create an XY plane through $Z = 0$ colored by *MixturePH*. The lower and upper bounds depend on the precision setting used in the ANSYS CFX-Solver should approximately range from 2 to 15 (single) or 2 to 11 (double).

Figure 1 shows two planes colored by `MixturePH`, with the plane on the right having a more accurate solution throughout the domain.

Figure 1 Comparison of Single and Double Precision Results for pH Variance



- View the acid, alkali and product mass fractions on the same plane.
- Create isosurfaces of Turbulence Kinetic Energy and Turbulence Eddy Dissipation.

Tutorial 14: Conjugate Heat Transfer in a Heating Coil

Introduction

This tutorial includes:

- [Tutorial 14 Features](#) (p. 240)
- [Overview of the Problem to Solve](#) (p. 241)
- [Defining a Simulation in ANSYS CFX-Pre](#) (p. 241)
- [Obtaining a Solution using ANSYS CFX-Solver Manager](#) (p. 246)
- [Viewing the Results in ANSYS CFX-Post](#) (p. 246)

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

- [Setting the Working Directory](#) (p. 1)
- [Changing the Display Colors](#) (p. 2)

Unless you plan on running a session file, you should copy the sample files used in this tutorial from the installation folder for your software (<CFXROOT>/examples/) to your working directory. This prevents you from overwriting source files provided with your installation. If you plan to use a session file, please refer to [Playing a Session File](#) (p. 241).

Sample files referenced by this tutorial include:

- **HeatingCoil.pre**
- **HeatingCoil_001.res**
- **HeatingCoilMesh.gtm**

Tutorial 14 Features

This tutorial addresses the following features of ANSYS CFX.

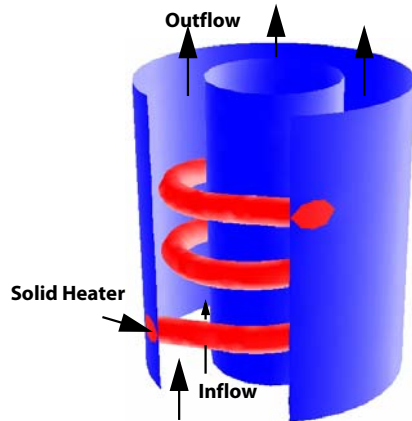
Component	Feature	Details	
ANSYS CFX-Pre	User Mode	General Mode	
	Simulation Type	Steady State	
	Fluid Type	General Fluid	
	Domain Type	Multiple Domain	
	Turbulence Model	k-Epsilon	
	Heat Transfer	Thermal Energy	
	Conjugate Heat Transfer		
	Subdomains	Energy Source	
	Boundary Conditions		Inlet (Subsonic)
			Opening
			Wall: No-Slip
			Wall: Adiabatic
	CEL (CFX Expression Language)		
Timestep	Physical Time Scale		
ANSYS CFX-Post	Plots	Cylinder	
		Default Locators	
		Isosurface	
	Other	Changing the Color Range	
		Expression Details View	
		Lighting Adjustment	
	Variable Details View		

In this tutorial you will learn about:

- Creating and using a solid domain as a heater coil in ANSYS CFX-Pre.
- Modeling conjugate heat transfer in ANSYS CFX-Pre.
- Specifying a subdomain to specify a heat source.
- Creating a cylinder locator using CEL in ANSYS CFX-Post.
- Examining the temperature distribution which is affected by heat transfer from the coil to the fluid.

Overview of the Problem to Solve

This example demonstrates the capability of ANSYS CFX in modeling conjugate heat transfer. In this example, part of the model of a simple heat exchanger is used to model the transfer of heat from a solid to a fluid. The model consists of a fluid domain and a solid domain. The fluid domain is an annular region through which water flows at a constant rate. The heater is a solid copper coil modeled as a constant heat source.



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 analysis in ANSYS. A results file is provided in case you wish to skip the model creation and solution steps within ANSYS CFX.

Defining a Simulation in ANSYS CFX-Pre

The following sections describe the simulation setup in ANSYS CFX-Pre.

Playing a Session File

If you wish to skip past these instructions, and have ANSYS CFX-Pre set up the simulation automatically, you can select **Session > Play Tutorial** from the menu in ANSYS CFX-Pre, then run the session file: `HeatingCoil.pre`. After you have played the session file as described in earlier tutorials under [Playing the Session File and Starting ANSYS CFX-Solver Manager](#) (p. 87), proceed to [Obtaining a Solution using ANSYS CFX-Solver Manager](#) (p. 246).

Creating a New Simulation

1. Start ANSYS CFX-Pre.
2. Select **File > New Simulation**.
3. Select **General** and click **OK**.
4. Select **File > Save Simulation As**.
5. Under **File name**, type `HeatingCoil`.

- Click **Save**.

Importing the Mesh

- Right-click `Mesh` and select **Import Mesh**.
- Apply the following settings

Setting	Value
File name	HeatingCoilMesh.gtm


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

Creating the Domains

This simulation requires both a fluid and a solid domain. First, you will create a fluid domain for the annular region of the heat exchanger.

Creating a Fluid Domain

The fluid domain will include the region of fluid flow but exclude the solid copper heater.

- Click *Domain*  and set the name to `FluidZone`.
- Apply the following settings to `FluidZone`

Tab	Setting	Value
General Options	Basic Settings > Location	B1.P3*
	Basic Settings > Fluids List	Water
	Domain Models > Pressure > Reference Pressure	1 [atm]
Fluid Models	Heat Transfer > Option	Thermal Energy
Initialization	Domain Initialization	(Selected)
	Domain Initialization > Initial Conditions	(Selected)
	Domain Initialization > Initial Conditions > Turbulence Eddy Dissipation	(Selected)

*. 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.

- Click **OK**.

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 a new domain named `SolidZone`.
- Apply the following settings

Tab	Setting	Value
General Options	Basic Settings > Location	B2.P3
	Basic Settings > Domain Type	Solid Domain
	Basic Settings > Solids List	Copper
Solid Models	Heat Transfer > Option	Thermal Energy
Initialization	Domain Initialization	(Selected)
	Domain Initialization > Initial Conditions	(Selected)
	Domain Initialization > Initial Conditions > Temperature > Option	Automatic with Value
	Domain Initialization > Initial Conditions > Temperature > Temperature	550 [K]

3. Click **OK**.

Creating a Subdomain to Specify a Thermal Energy Source

To allow a thermal energy source to be specified for the copper heating element, you need to create a subdomain.

1. Create a new subdomain named `Heater` in the domain `SolidZone`.
2. Apply the following settings

Tab	Setting	Value
Basic Settings	Basic Settings > Location	B2.P3*
Sources	Sources	(Selected)
	Sources > Equation Sources > Energy	(Selected)
	Sources > Equation Sources > Energy > Source	1.0E+07 [W m ⁻³]

*. This is the same location as for the domain `SolidZone`, because you want the source term to apply to the entire solid domain.

3. Click **OK**.

Creating the Boundary Conditions

Inlet Boundary You will now create an inlet boundary condition for the cooling fluid (Water).

1. Create a new boundary condition named `inflow` in the domain `FluidZone`.
2. Apply the following settings

Tab	Setting	Value
Basic Settings	Boundary Type	Inlet
	Location	inflow
Boundary Details	Mass and Momentum > Normal Speed	0.4 [m s ⁻¹]
	Heat Transfer > Static Temperature	300 [K]

3. Click **OK**.

Opening boundary

The opening boundary condition type is used 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. Create a new expression named `OutletTemperature`.
2. Set **Definition** to `areaAve(T)@REGION:outflow`
3. Click **Apply**.
4. Create a new boundary condition named `outflow` in the domain **FluidZone**.
5. 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

6. Click **OK**.

The default adiabatic wall boundary condition will automatically be applied to the remaining unspecified external boundaries of the fluid domain. The default Fluid-Solid Interface boundary condition (flux conserved) will be applied to the surfaces between the solid domain and the fluid domain.

Creating the Domain Interface

If you have the **Generate Default Domain Interfaces** option turned on (from **Edit > Options > CFX-Pre**), then you will see that an interface called `Default Fluid Solid Interface` already exists, and is listed in the **Outline** tree view. If this is the case, you can optionally skip the following instructions for creating a domain interface (since the domain interface set here will have the same settings as, and will automatically replace, the default domain interface).

If you have the **Generate Default Domain Interfaces** option turned off, then there is no domain interface defined at this point. In this case, create a domain interface using *either one* of the following methods (the result is the same):

Creating a Default Domain Interface

1. Right click `Simulation` in the **Outline** tree view and ensure that **Automatic Default Interfaces** is selected. An interface named `Default Fluid Solid Interface` should now appear under the `Simulation` branch.


Creating a Domain Interface Manually

1. Double click `Default Fluid Solid Interface` and apply the following settings:

Tab	Setting	Value
Basic Settings	Interface Type	Fluid Solid
	Interface Side 1 > Domain (Filter)	FluidZone
	Interface Side 1 > Region List	F10.B1.P3, F5.B1.P3, F6.B1.P3, F7.B1.P3, F8.B1.P3, F9.B1.P3
	Interface Side 2 > Domain (Filter)	SolidZone
	Interface Side 2 > Region List	F10.B2.P3, F5.B2.P3, F6.B2.P3, F7.B2.P3, F8.B2.P3, F9.B2.P3
	Interface Models > Option	General Connection
	Interface Models > Frame Change/Mixing Model > Option	None
	Interface Models > Pitch Change > Option	None
	Mesh Connection Method > Option	Automatic

2. Click **OK**.

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**.

Writing the Solver (.def) File

1. Click *Write Solver File* .
2. Apply the following settings:

Setting	Value
File name	HeatingCoil.def
Quit CFX-Pre *	(Selected)

*. If using ANSYS CFX-Pre in Standalone Mode.

3. Ensure **Start Solver Manager** is selected and click **Save**.

4. If using Standalone Mode, quit ANSYS CFX-Pre, saving the simulation (.cfx) file at your discretion.

Obtaining a Solution using ANSYS 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., **Heat Transfer, 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 **Define Run** is displayed.
2. Click **Start Run**.
ANSYS 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. Click **Yes** to post-process the results.
4. If using Standalone Mode, quit ANSYS CFX-Solver Manager.

Viewing the Results in ANSYS CFX-Post

The following topics will be discussed:

- [Creating a Cylindrical Locator](#) (p. 246)
- [Specular Lighting](#) (p. 247)
- [Moving the Light Source](#) (p. 247)

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.

Expression

1. Create a new expression named `expradius`.
2. Apply the following settings

Setting	Value
Definition	$(x^2 + y^2)^{0.5}$

3. Click **Apply**.

Variable

1. Create a new variable named `radius`.
2. Apply the following settings

Setting	Value
Expression	<code>expradius</code>

3. Click **Apply**.

Isosurface of the variable

1. Create a new isosurface named `Isosurface 1`.
2. Apply the following settings

Tab	Setting	Value
Geometry	Definition > Variable	radius
	Definition > Value	0.8 [m]*
Color	Mode	Variable
	Variable	Temperature
	Range	User Specified
	Min	300 [K]
	Max	302 [K]
Render	Draw Faces	(Selected)

*. The maximum radius is 1 m, so a cylinder locator at a radius of 0.8 m is suitable.

3. Click **Apply**.

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.

1. Apply the following settings to `Isosurface 1`

Tab	Setting	Value
Render	Draw Faces > Specular	(Cleared)

2. Click **Apply**.

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.

Exporting the Results to ANSYS

This optional step involves generating an ANSYS `.cdb` data file from the results generated in ANSYS 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 ANSYS CFX-Solver Manager to export data. For details, see [Exporting Data from ANSYS CFX-Solver Manager](#) (p. 248).
- Use ANSYS CFX-Post to export data. This involves:
 - a. Importing a surface mesh from ANSYS into ANSYS CFX-Post, and associating the surface with the corresponding 2D region in the ANSYS CFX-Solver results file.
 - b. Exporting the data to a file containing SFE commands that represent surface element thermal or mechanical stress values.
 - c. Loading the commands created in the previous step into ANSYS and visualizing the loads.

Exporting Data from ANSYS CFX-Solver Manager

Since the heat transfer in the solid domain was calculated in ANSYS CFX, the 3D thermal data will be exported to **ANSYS Element Type** as `3D Thermal (70)` data. The mechanical stresses are calculated on the liquid side of the liquid-solid interface. These values will be exported to **ANSYS Element Type** as `2D Stress (154)` data.

Thermal Data

1. Start ANSYS CFX-Solver Manager.
2. Select **Tools > Export to ANSYS MultiField**.
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.csv
Domain Name > Domain	SolidZone
Domain Name > Boundary*	*
Export Options > ANSYS Element Type	3D Thermal (70)

*. Leave Boundary empty.

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](#).

Mechanical Stresses

1. Apply the following settings in the **Export to ANSYS MultiField Solver** dialog box (see [Step 2](#) above):

Setting	Value
Results File	HeatingCoil_001.res
Export File	HeatingCoil_001_ansysfsi_154.csv
Domain Name > Domain	FluidZone
Domain Name > Boundary	FluidZone Default
Export Options > ANSYS Element Type	2D Stress (154)

2. Click **Export**.

You now have two exported files that can be loaded into ANSYS Multiphysics. When you are finished, close ANSYS CFX-Solver Manager and ANSYS CFX-Post.

Tutorial 15: Multiphase Flow in Mixing Vessel

Introduction

This tutorial includes:

- [Tutorial 15 Features](#) (p. 252)
- [Overview of the Problem to Solve](#) (p. 253)
- [Defining a Simulation in ANSYS CFX-Pre](#) (p. 253)
- [Obtaining a Solution using ANSYS CFX-Solver Manager](#) (p. 265)
- [Viewing the Results in ANSYS CFX-Post](#) (p. 265)

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

- [Setting the Working Directory](#) (p. 1)
- [Changing the Display Colors](#) (p. 2)

Unless you plan on running a session file, you should copy the sample files used in this tutorial from the installation folder for your software (<CFXROOT>/examples/) to your working directory. This prevents you from overwriting source files provided with your installation. If you plan to use a session file, please refer to [Playing a Session File](#) (p. 253).

Sample files referenced by this tutorial include:

- **MixerImpellerMesh.gtm**
- **MixerTank.geo**
- **MultiphaseMixer.pre**

Tutorial 15 Features

This tutorial addresses the following features of ANSYS CFX.

Component	Feature	Details	
ANSYS CFX-Pre	User Mode	General Mode	
	Simulation Type	Steady State	
	Fluid Type	General Fluid	
	Domain Type		Multiple Domain
			Rotating Frame of Reference
	Turbulence Model		Dispersed Phase Zero Equation
			Fluid-Dependant Turbulence Model
			k-Epsilon
	Heat Transfer	None	
	Buoyant Flow		
	Multiphase		
	Boundary Conditions		Inlet (Subsonic)
			Outlet (Degassing)
			Wall: Thin Surface
Wall: (Slip Depends on Volume Fraction)			
Domain Interfaces		Frozen Rotor	
		Periodic	
Output Control			
Timestep		Physical Time Scale	
ANSYS CFX-Post	Plots	Default Locators	
		Isosurface	
		Slice Plane	
	Other	Quantitative Calculation	

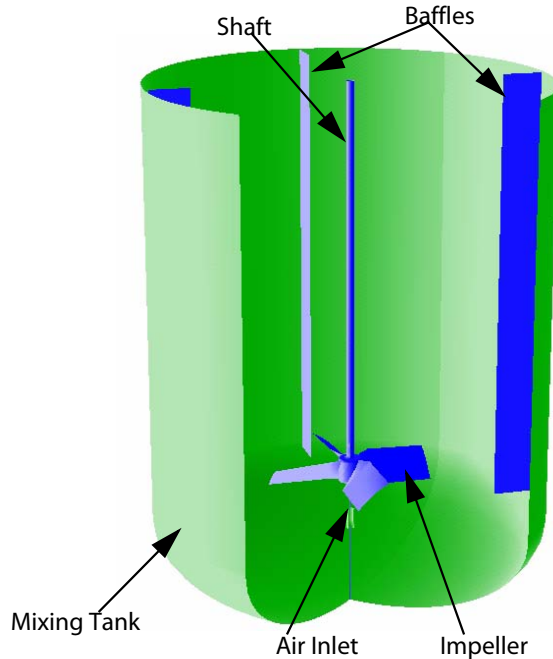
In this tutorial you will learn about:

- Importing meshes that have CFX-4 and ANSYS CFX `.def/.res` file formats.
- Setting up a simulation using multiple frames of reference.
- Connecting two domains (one for the impeller and one for the tank) via Frozen Rotor interfaces.
- Modeling rotational periodicity using periodic boundary conditions.
- Using periodic GGI interfaces where the mesh does not map exactly.
- Using thin surfaces for the blade and baffle surfaces.
- Setting up a multiphase flow problem.

Overview of the Problem to Solve

This example simulates the mixing of two fluids in a mixing vessel. The geometry consists of a mixing tank vessel containing four baffles. A rotating impeller blade is connected to a shaft which runs vertically through the vessel. Air is injected into the vessel through an inlet pipe located below the impeller blade at a speed of 5 m/s.

Figure 1 Cut-away diagram of Mixing Vessel



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.

Defining a Simulation in ANSYS CFX-Pre

The following sections describe the simulation setup in ANSYS CFX-Pre.

Playing a Session File

If you wish to skip past these instructions, and have ANSYS CFX-Pre set up the simulation automatically, you can select **Session > Play Tutorial** from the menu in ANSYS CFX-Pre, then run the session file: **MultiphaseMixer.pre**. After you have played the session file as described in earlier tutorials under [Playing the Session File and Starting ANSYS CFX-Solver Manager](#) (p. 87), proceed to [Obtaining a Solution using ANSYS CFX-Solver Manager](#) (p. 265).

Creating a New Simulation

1. Start ANSYS CFX-Pre.

2. Select **File > New Simulation**.
3. Select **General** and click **OK**.
4. Select **File > Save Simulation As**.
5. Under **File name**, type `MultiphaseMixer`.
6. Click **Save**.

Importing the Meshes

In this tutorial, a CFX-4 mesh is imported using advanced options. These options control how the CFX-4 mesh is imported into ANSYS CFX.

By creating 3D regions on fluid regions, you prevent import of `USER3D` and `POROUS` regions. Turn off this option if you do not need these regions for sub-domains. This will simplify the regions available in ANSYS CFX-Pre. In this case, the mesh file contains `USER3D` regions that were created as a location for a thin surface and you do not need them for defining any subdomains.

Importing the Mixer Tank Mesh

1. Right-click `Mesh` and select **Import Mesh**.
2. Apply the following settings

Setting	Value
File type	CFX-4 (*.geo)
File name	MixerTank.geo
Advanced Options > Create 3D Regions on > Fluid Regions (USER3D, POROUS)	(Cleared)

3. Click **Open**.

Importing the Impeller Mesh

1. Right-click `Mesh` and select **Import Mesh** to import the second mesh.
2. Apply the following settings

Setting	Value
File type	CFX Mesh (*.gtm)
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.

Transforming 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

Tab	Setting	Value
Definition	Transformation	Translation
	Apply Translation > Method	Deltas
	Apply Translation > Dx, Dy, Dz	0.275, 0, 0

3. Click **OK**.


Viewing the Mesh at the Tank Periodic Boundary

1. Click *Label and Marker Visibility* .
2. Apply the following setting

Tab	Setting	Value
Label Options	Show Labels	(Cleared)


3. Click **OK**.
4. In the **Outline** workspace, expand `MixerImpellerMesh.gtm` and `MixerTank.geo` to view associated 2D primitives.
5. Under `MixerTank.geo > Principal 3D regions > Primitive 3D`, 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 *Highlight* . 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 > Viewer**.

Creating the Domains

Rotating Domain for the Impeller

1. Click *Domain*  and set the name to `impeller`.
2. Apply the following settings

Tab	Setting	Value
General Options	Basic Settings > Location	Main
	Basic Settings > Fluids List	Air at 25 C, 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 *	997 [kg m ⁻³]
	Domain Models > Domain Motion > Option	Rotating
	Domain Models > Domain Motion > Angular Velocity	84 [rev min ⁻¹] [†]
	Domain Models > Domain Motion > Axis Definition > Rotation Axis	Global X
Fluid Models	Multiphase Options > Homogeneous Model	(Cleared)
	Multiphase Options > Allow Musig Fluids	(Cleared)
	Multiphase Options > 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 Details	Fluid Details	Air at 25 C
	Fluid Details > Air at 25 C > Morphology > Option	Dispersed Fluid
	Fluid Details > Air at 25 C > Morphology > Mean Diameter	3 [mm]
Fluid Pairs	Fluid Pairs	Air at 25 C Water
	Fluid Pairs > Air at 25 C Water > Surface Tension Coefficient	(Selected)
	Fluid Pairs > Air at 25 C Water > Surface Tension Coefficient > Surf. Tension Coeff.	0.073 [N m ⁻¹] [‡]
	Fluid Pairs > Air at 25 C Water > Momentum Transfer > Drag Force > Option	Grace
	Fluid Pairs > Air at 25 C Water > Momentum Transfer > Drag Force > Volume Fraction Correction Exponent	(Selected)
	Fluid Pairs > Air at 25 C Water > Momentum Transfer > Drag Force > Volume Fraction Correction Exponent > Value	4
	Fluid Pairs > Air at 25 C Water > Momentum Transfer > Non-drag forces > Turbulent Dispersion Force > Option	Lopez de Bertodano
	Fluid Pairs > Air at 25 C Water > Momentum Transfer > Non-drag forces > Turbulent Dispersion Force > Dispersion Coeff.	0.1
	Fluid Pairs > Air at 25 C Water > Turbulence Transfer > Option	Sato Enhanced Eddy Viscosity

- *. For dilute dispersed multiphase flow, always set the buoyancy reference density to that for continuous fluid.
- †. Note the unit.
- ‡. This must be set to allow the Grace drag model to be used.

3. Click **OK**.

Stationary Domain for the Main Tank

Next, you will create a stationary domain for the main tank by copying the properties of the existing fluid domain.

1. Right-click `impeller` and select **Duplicate** from the shortcut menu.
2. Set the name of this domain to `tank` and open it for editing.
3. Apply the following settings

Tab	Setting	Value
General Options	Basic Settings > Location	Primitive 3D
	Domain Models > Domain Motion > Option	Stationary

4. Click **OK**.

Creating the Boundary Conditions

The following boundary conditions that define the problem 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 and impeller blade.
- A wall for the hub and shaft in the rotating domain. This will be stationary relative to the rotating domain.
- A wall for the shaft in the stationary domain. This will be rotating relative to the stationary domain.
- Periodic domain interfaces for the periodic faces of the tank and impeller.

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.

When the default wall boundary condition is generated, the internal 2D regions of an imported mesh are ignored, while the regions that form domain boundaries are included.

Air Inlet Boundary

1. Create a new boundary condition 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

Tab	Setting	Value
Fluid Values	Boundary Conditions	Air at 25 C
	Boundary Conditions > Air at 25 C > Velocity > Normal Speed	5 [m s ⁻¹]
	Boundary Conditions > Air at 25 C > Volume Fraction > Volume Fraction	1
	Boundary Conditions	Water
	Boundary Conditions > Water > Velocity > Normal Speed	5 [m s ⁻¹]
	Boundary Conditions > Water > Volume Fraction > Volume Fraction	0

3. Click **OK**.

Degassing Outlet Boundary

1. Create a new boundary condition 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**.

Thin Surface for the Baffle

In ANSYS 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 condition in the domain `tank` named `Baffle`.
2. Apply the following settings

Tab	Setting	Value
Basic Settings	Boundary Type	Wall
	Location	WALL_BAFFLES*
Boundary Details	Wall Influence On Flow > Option	Fluid Dependent
Fluid Values	Boundary Conditions	Air at 25 C
	Boundary Conditions > Air at 25 C > Wall Influence on Flow > Option	Free Slip
	Boundary Conditions	Water
	Boundary Conditions > Water > Wall Influence on Flow > Option	No Slip [†]

*. 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). Therefore, you do not need to use the **Create Thin Surface Partner** option.

†. 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.

3. Click **OK**.

Wall Boundary Condition for the Shaft

The next stage involves setting up a boundary condition for the shaft, which exists in the tank (stationary domain). These regions are connected to the shaft in the impeller domain. Since the tank domain is not rotating, you need to specify a moving wall to account for the rotation of the shaft.

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 condition 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 condition 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	Wall Influence On Flow > Option	Fluid Dependent
Fluid Values	Boundary Conditions	Air at 25 C
	Boundary Conditions > Air at 25 C > Wall Influence on Flow > Option	No Slip
	Boundary Conditions > Air at 25 C > Wall Influence on Flow > Wall Velocity	(Selected)
	Boundary Conditions > Air at 25 C > Wall Influence on Flow > Wall Velocity > Option	Rotating Wall
	Boundary Conditions > Air at 25 C > Wall Influence on Flow > Wall Velocity > Angular Velocity	84 [rev min-1]*
	Boundary Conditions > Air at 25 C > Wall Influence on Flow > Wall Velocity > Axis Definition > Rotation Axis	Global X

*. Note the unit.

3. Select `Water` and set the same values as for `Air at 25 C`.
4. Click **OK**.

Required Boundary Conditions in the Impeller Domain

1. Create a new boundary condition in the domain `impeller` named `Blade`.
2. Apply the following settings

Tab	Setting	Value
Basic Settings	Boundary Type	Wall
	Location	Blade
	Thin Surfaces > Create Thin Surface Partner	(Selected)*
Boundary Details	Wall Influence On Flow > Option	Fluid Dependent
Fluid Values	Boundary Conditions	Air at 25 C
	Boundary Conditions > Air at 25 C > Wall Influence on Flow > Option	Free Slip
	Boundary Conditions	Water
	Boundary Conditions > Water > Wall Influence on Flow > Option	No Slip

*. The `Blade` region only includes the surface from one side of the blade (you can confirm this by examining `Blade` in the region selector). Therefore, you can select **Create Thin Surface Partner** to include the surfaces from the other side of the blade.

3. Click **OK**.
You will see in the tree view that a boundary named `Blade Other Side` has automatically been created.
4. Create a new boundary condition in the domain `impeller` named `HubShaft`.
5. Apply the following settings

Tab	Setting	Value
Basic Settings	Boundary Type	Wall
	Location	Hub, Shaft
Boundary Details	Wall Influence On Flow > Option	Fluid Dependent
Fluid Values	Boundary Conditions	Air at 25 C
	Boundary Conditions > Air at 25 C > Wall Influence on Flow > Option	Free Slip
	Boundary Conditions	Water
	Boundary Conditions > Water > Wall Influence on Flow > Option	No Slip

6. Click **OK**.
1. On the tree view, open `tank Default` for editing.
2. Apply the following settings

Modifying the Default Wall Boundary Condition

Tab	Setting	Value
Boundary Details	Wall Influence On Flow > Option	Fluid Dependent

Tab	Setting	Value
Fluid Values	Boundary Conditions	Air at 25 C
	Boundary Conditions > Air at 25 C > Wall Influence on Flow > Option	Free Slip
	Boundary Conditions	Water
	Boundary Conditions > Water > Wall Influence on Flow > Option	No Slip

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.

Creating the Domain Interfaces

Impeller Domain

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 > Rotation Axis	Global X

3. Click **OK**.

Tank Domain

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 > Rotation Axis	Global X

3. Click **OK**.

Frozen Rotor Interface

Next, you will 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 > 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 > 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
	Interface Models > Frame Change/Mixing Model > Option	Frozen Rotor

9. Click **OK**.

Setting Initial Values

The initialization for volume fraction is 0 for air and automatic for water. Therefore, the initial volume fraction for water will be set to 1 so that the sum of the two fluid volume fractions is 1.

It is important to understand how the velocity is initialized in this tutorial. Here, both fluids use `Automatic` for the **Cartesian Velocity Components**. 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 at 25 C
	Fluid Specific Initialization > Air at 25 C > Initial Conditions > Volume Fraction > Option	Automatic with Value
	Fluid Specific Initialization > Air at 25 C > Initial Conditions > Volume Fraction > Volume Fraction	0
	Fluid Specific Initialization	Water
	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]
	Fluid Specific Initialization > Water > Initial Conditions > Turbulence Eddy Dissipation	(Selected)

3. Click **OK**.

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 timestep 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 timestep 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 timestep.

Note: You may find it useful to 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. You could create the following expressions:

```
TankAirHoldUp = volumeAve(Air at 25 C.vf)@tank
ImpellerAirHoldUp = volumeAve(Air at 25 C.vf)@impeller
TotalAirHoldUp = (volume()@tank * TankAirHoldUp + volume()@impeller *
ImpellerAirHoldUp) / (volume()@tank + volume()@impeller)
```

and then monitor the value of TotalAirHoldUp.

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]*

*. This is an aggressive timestep for this case.

3. Click **OK**.

Setting Output Control

In the next step, you will choose to write additional data to the results file which allows force and torque calculations to be performed in post-processing.


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

Tab	Setting	Value
Results	Output Boundary Flows	(Selected)
	Output Boundary Flows > Boundary Flows	All

3. Click **OK**.

Writing the Solver (.def) File

Since this tutorial uses domain interfaces and you choose to summarize the interface data, an information window is displayed that informs you of the connection type used for each domain interface.

1. Click *Write Solver File* .
2. Apply the following settings:

Setting	Value
File name	MultiphaseMixer.def
Summarize Interface Data	(Selected)
Quit CFX-Pre*	(Cleared)

*. If using ANSYS CFX-Pre in Standalone Mode.

3. Ensure **Start Solver Manager** is selected and click **Save**.
If you are notified the file already exists, click **Overwrite**.
A message about interface connections appears.
4. Click **OK**.
5. If using Standalone Mode, quit ANSYS CFX-Pre, saving the simulation (.cfx) file at your discretion.

Obtaining a Solution using ANSYS CFX-Solver Manager

The ANSYS CFX-Solver Manager will be launched after ANSYS CFX-Pre has closed down. 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**.
ANSYS CFX-Solver runs and attempts to obtain a solution. This can take a long time depending on your system.
After a run has finished, examine some of the information printed at the end of the OUT file.
A common quantity of interest is the mass balance; this compares the amount of fluid leaving the domain to the amount entering.
 - You usually want the **Global Imbalance, in %**: to be less than 0.1 % in a converged solution.
 - For a single phase calculation, the mass balance is the **P-Mass** equation.
 - For a multiphase calculation, examine the information given for the **P-Vol** equation.
 - This is not the volumetric flow balance information, but is the summation of the phasic continuity mass balance information.
3. Click **Yes** to post-process the results when the completion message appears at the end of the run.
4. If using Standalone Mode, quit ANSYS CFX-Solver Manager.

Viewing the Results in ANSYS CFX-Post

When ANSYS CFX-Post has started you will be able to see the mixer geometry in the Viewer. You will create some plots showing how effective mixing has occurred. You will also calculate the torque and power required by the impeller.

Visualizing the Mixing Process

- Creating a plane**
1. Create a new plane named `Plane 1`.
 2. 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
Color	Mode	Variable
	Variable	Air at 25 C.Volume Fraction
	Range	User Specified
	Min	0
	Max	0.04

3. Click **Apply**.
4. Observe the plane, then apply the following settings:

Tab	Setting	Value
Color	Variable	Air at 25 C.Shear Strain Rate
	Range	User Specified
	Min	0 [s ⁻¹]
	Max	15 [s ⁻¹]

5. Click **Apply**.
Areas of high shear strain rate or shear stress are typically also areas where the highest mixing occurs.
6. Observe the plane, then apply the following settings:

Tab	Setting	Value
Color	Variable	Pressure
	Range	Local

7. Click **Apply**.
Note that the hydrostatic contribution to pressure is excluded due to the use of an appropriate buoyancy reference density. If you plot the variable called `Absolute Pressure`, you will see the true pressure including the hydrostatic contribution.

Creating a vector

1. Create a new vector named `Vector 1`.
2. Apply the following settings

Tab	Setting	Value
Geometry	Definition > Locations	Plane 1
	Variable	Water.Velocity in Stn Frame *
Symbol	Symbol Size	0.2
	Normalize Symbols	(Selected)

*. 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. Observe the vector plot, then change the variable to `Air at 25 C.Velocity in Stn Frame`. Observe this as well, then clear the visibility of `Vector 1`.
4. Modify the `tank Default` object.
5. Apply the following settings:

Tab	Setting	Value
Color	Mode	Variable
	Variable	Water.Wall Shear
	Range	Local

The legend for this plot shows the range of wall shear values.

The global maximum wall shear 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 will be small since the contact area fraction of water here is very small.

Calculating Power and Torque Required by the Impeller

1. Select **Tools > Function Calculator** from the main menu or click *Show Function Calculator*  from the main toolbar.
2. Apply the following settings:

Tab	Setting	Value
Function Calculator	Function	torque
	Location	Blade
	Axis	Global X
	Fluid	All Fluids

3. Click **Calculate** to find the torque required to rotate `Blade` about the X-axis.
4. Repeat the calculation setting **Location** to `Blade Other Side`.

The sum of these two results is the torque required by the single impeller blade, approximately 70 [N m]. This must be multiplied by the number of blades in the full geometry to obtain the total torque required by the impeller; the result is a value of approximately 282 [N m]. You could also include the results 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 (8.8 rad/s): $\text{Power} = 282 * 8.8 = 2482 \text{ [W]}$.

Remember that this value is the power requirement for the work done on the fluid only, it does not account for any mechanical losses, 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.

Tutorial 16: Gas-Liquid Flow in an Airlift Reactor

Introduction

This tutorial includes:

- [Tutorial 16 Features](#) (p. 270)
- [Overview of the Problem to Solve](#) (p. 270)
- [Defining a Simulation in ANSYS CFX-Pre](#) (p. 271)
- [Obtaining a Solution using ANSYS CFX-Solver Manager](#) (p. 277)
- [Viewing the Results in ANSYS CFX-Post](#) (p. 278)
- [Additional Fine Mesh Simulation Results](#) (p. 280)

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

- [Setting the Working Directory](#) (p. 1)
- [Changing the Display Colors](#) (p. 2)

Unless you plan on running a session file, you should copy the sample files used in this tutorial from the installation folder for your software (<CFXROOT>/examples/) to your working directory. This prevents you from overwriting source files provided with your installation. If you plan to use a session file, please refer to [Playing a Session File](#) (p. 271).

Sample files referenced by this tutorial include:

- **BubbleColumn.pre**
- **BubbleColumnMesh.gtm**

Tutorial 16 Features

This tutorial addresses the following features of ANSYS CFX.

Component	Feature	Details	
ANSYS CFX-Pre	User Mode	General	
	Simulation 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: Thin Surface	
		Wall: (Slip Depends on Volume Fraction)	
Timestep	Physical Time Scale		
ANSYS CFX-Post	Plots	Default Locators	
		Vector	
	Other	Changing the Color Range	
	Symmetry		

In this tutorial you will learn about:

- Setting up a multiphase flow 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 condition to allow air, but not water, to escape from the boundary.

Overview of the Problem to Solve

This tutorial demonstrates the Eulerian–Eulerian multiphase model in ANSYS CFX. The tutorial simulates a bubble column with an internal tube (draft tube) used to direct recirculation of the flow. This configuration is known as an airlift reactor. Bubble columns are tall gas-liquid contacting vessels and are often used in processes where gas absorption is important (e.g., bioreactors to dissolve oxygen in broths) and to limit the exposure of micro-organisms to excessive shear, imparted by mechanically driven mixers.

This example models the dispersion of air bubbles in water. The gas is supplied through a sparger at the bottom of the vessel and the rising action of the bubbles provides gentle agitation of the liquid.

Simple bubble columns that are without the draft tube tend to develop irregular flow patterns and poor overall mixing. The draft tube in the airlift reactor helps establish a regular flow pattern in the column and achieve better uniformity of temperature, concentration and pH in the liquid phase, but sometimes at the expense of decreased mass transfer from the gas to the liquid.

This tutorial also demonstrates the use of thin surfaces. Thin surfaces are internal two dimensional wall boundaries used to model thin three dimensional features (e.g., baffles, guide vanes within ducts, etc.).

The airlift reactor that is modeled here is very similar to the laboratory bench scale prototype used by García-Calvo and Letón.

If you are interested, 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. 280).

Defining a Simulation in ANSYS CFX-Pre

The following sections describe the simulation setup in ANSYS CFX-Pre.

Playing a Session File

If you wish to skip past these instructions, and have ANSYS CFX-Pre set up the simulation automatically, you can select **Session > Play Tutorial** from the menu in ANSYS CFX-Pre, then run the session file: `BubbleColumn.pre`. After you have played the session file as described in earlier tutorials under [Playing the Session File and Starting ANSYS CFX-Solver Manager](#) (p. 87), proceed to [Obtaining a Solution using ANSYS CFX-Solver Manager](#) (p. 277).

Creating a New Simulation

1. Start ANSYS CFX-Pre.
2. Select **File > New Simulation**.
3. Select **General** and click **OK**.
4. Select **File > Save Simulation As**.
5. Under **File name**, type `BubbleColumn`.
6. Click **Save**.

Importing the Mesh

1. Right-click `Mesh` and select **Import Mesh**.
2. Apply the following settings

Setting	Value
File name	BubbleColumnMesh.gtm

3. Click **Open**.

Creating the Domain

1. Right click `Simulation` in the **Outline** tree view and ensure that **Automatic Default Domain** is selected. 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
General Options	Basic Settings > Location	B1.P3, B2.P3
	Basic Settings > Fluids List	Air at 25 C, 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*	997 [kg m ⁻³]
Fluid Models	Multiphase Options > Homogeneous Model	(Cleared)
	Multiphase Options > Allow Musig Fluids	(Cleared)
	Free Surface Model > Option	None
	Heat Transfer > Option	Isothermal
	Heat Transfer > Fluid Temperature	25 C
	Turbulence > Option	Fluid Dependent
Fluid Details	Fluid Details	Air at 25 C
	Fluid Details > Air at 25 C > Morphology > Option	Dispersed Fluid
	Fluid Details > Air at 25 C > Morphology > Mean Diameter	6 [mm]

Tab	Setting	Value
Fluid Pairs	Fluid Pairs > Air at 25 C Water > Surface Tension Coefficient	(Selected)
	Fluid Pairs > Air at 25 C Water > Surface Tension Coefficient > Surf. Tension Coeff.	0.072 [N m ⁻¹] [†]
	Fluid Pairs > Air at 25 C Water > Momentum Transfer > Drag Force > Option	Grace
	Fluid Pairs > Air at 25 C Water > Momentum Transfer > Drag Force > Volume Fraction Correction Exponent	(Selected)
	Fluid Pairs > Air at 25 C Water > Momentum Transfer > Drag Force > Volume Fraction Correction Exponent > Value	2
	Fluid Pairs > Air at 25 C Water > Momentum Transfer > Non-drag Forces > Turbulent Dispersion Force > Option	Lopez de Bertodano
	Fluid Pairs > Air at 25 C Water > Momentum Transfer > Non-drag Forces > Turbulent Dispersion Force > Dispersion Coeff.	0.3
	Fluid Pairs > Air at 25 C Water > Turbulence Transfer > Option	Sato Enhanced Eddy Viscosity

- *. For dilute dispersed multiphase flow, always set the buoyancy reference density to that for continuous fluid.
- †. This must be set to allow the Grace drag model to be used.

3. Click **OK**.

Creating the Boundary Conditions

For this simulation of the airlift reactor, the boundary conditions required are:

- An inlet for air on the sparger.
- A degassing outlet for air at the liquid surface.
- A thin surface wall for the draft tube.
- An exterior wall for the outer wall, base and sparger tube.
- Symmetry planes for the cross sections.

Inlet Boundary

There are an infinite number of inlet velocity/volume fraction combinations that will produce the same mass inflow of air. The combination chosen gives an air inlet velocity close to the terminal rise velocity. Since the water inlet velocity is zero, you can adjust its volume fraction until the required mass flow rate of air is obtained for a given air inlet velocity.

1. Create a new boundary condition 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

Tab	Setting	Value
Fluid Values	Boundary Conditions	Air at 25 C
	Boundary Conditions > Air at 25 C > Velocity > Normal Speed	0.3 [m s ⁻¹]
	Boundary Conditions > Air at 25 C > Volume Fraction > Volume Fraction	0.25
	Boundary Conditions	Water
	Boundary Conditions > Water > Velocity > Normal Speed	0 [m s ⁻¹]
	Boundary Conditions > Water > Volume Fraction > Volume Fraction	0.75

3. Click **OK**.

Outlet Boundary

The top of the reactor will be a degassing boundary, which is classified as an outlet boundary.

1. Create a new boundary condition named T_{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**.

Thin Surface Draft Tube Boundary

Thin surfaces are created by specifying a wall boundary condition on *both* sides of an internal region. If only one side has a boundary condition then the ANSYS CFX-Solver will fail. To assist with this, you can select only one side of a thin surface and then enable the **Create Thin Surface Partner** toggle. ANSYS CFX-Pre will then try to automatically create another boundary condition for the other side.

1. Create a new boundary condition named $DraftTube$.
2. Apply the following settings

Tab	Setting	Value
Basic Settings	Boundary Type	Wall
	Location	Draft Tube
	Thin Surfaces > Create Thin Surface Partner	(Selected)
Boundary Details	Wall Influence On Flow > Option	Fluid Dependent
Fluid Values	Boundary Conditions	Air at 25 C
	Boundary Conditions > Air at 25 C > Wall Influence On Flow > Option	Free Slip
	Boundary Conditions	Water
	Boundary Conditions > Water > Wall Influence On Flow > Option	No Slip

3. Click **OK**.

A boundary condition named `DraftTube Other Side` will now be created automatically.

Symmetry Plane Boundary

In this step you will create symmetry plane boundary conditions on the `Symmetry1` and `Symmetry2` locators, one for each of the two vertical cross sections of the reactor sector.

1. Create a new boundary condition 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 condition named `SymP2`.
5. Apply the following settings

Tab	Setting	Value
Basic Settings	Boundary Type	Symmetry
	Location	Symmetry2

6. Click **OK**.

Modifying the Default Boundary

The remaining external regions are assigned to the default wall boundary condition. This needs to be modified to set the `Air` phase to `Free Slip`.

1. In the **Outline** workspace, open `Default Domain Default` for editing.
2. Apply the following settings

Tab	Setting	Value
Boundary Details	Wall Influence on Flow > Option	Fluid Dependent
Fluid Values	Boundary Conditions	Air at 25 C
	Boundary Conditions > Air at 25 C > Wall Influence on Flow > Option	Free Slip


3. Click **OK**.

The boundary condition specifications are now complete.

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 bubble column problems, improved convergence can be obtained by using CEL (CFX Expression Language) to specify a non zero volume fraction, for air in the riser and a zero value in the downcomer. This should be done if two solutions are possible (for example, if the flow could go up the downcomer and down the riser).

1. Click *Global Initialization* 

Since a single pressure field exists for a multiphase calculation you do not set pressure values on a per fluid basis.

2. Apply the following settings


Tab	Setting	Value
Fluid Settings	Fluid Specific Initialization	Air at 25 C
	Fluid Specific Initialization > Air at 25 C	(Selected)
	Fluid Specific Initialization > Air at 25 C > Initial Conditions > Cartesian Velocity Components > Option	Automatic with Value
	Fluid Specific Initialization > Air at 25 C > Initial Conditions > Cartesian Velocity Components > U	0 [m s ⁻¹]
	Fluid Specific Initialization > Air at 25 C > Initial Conditions > Cartesian Velocity Components > V	0.3 [m s ⁻¹]
	Fluid Specific Initialization > Air at 25 C > Initial Conditions > Cartesian Velocity Components > W	0 [m s ⁻¹]
	Fluid Specific Initialization	Water*
	Fluid Specific Initialization > Water	(Selected)
	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 > Turbulence Kinetic Energy > Option	Automatic
	Fluid Specific Initialization > Water > Initial Conditions > Turbulence Eddy Dissipation	(Selected)
	Fluid Specific Initialization > Water > Initial Conditions > Turbulence Eddy Dissipation > Option	Automatic
	Fluid Specific Initialization > Water > Initial Conditions > Volume Fraction > Option	Automatic with Value
	Fluid Specific Initialization > Water > Initial Conditions > Volume Fraction > Volume Fraction	1 [†]

- *. Since there is no water entering or leaving the domain, a stationary initial guess is recommended.
- †. 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**.

Setting Solver Control


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.

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	1 [s]

3. Click **OK**.

Writing the Solver (.def) File

1. Click *Write Solver File* .
2. Apply the following settings:

Setting	Value
File name	BubbleColumn.def
Quit CFX-Pre*	(Selected)

*. If using ANSYS CFX-Pre in Standalone Mode.

3. Ensure **Start Solver Manager** is selected and click **Save**.
4. If using Standalone Mode, quit ANSYS CFX-Pre, saving the simulation (.cfx) file at your discretion.

Obtaining a Solution using ANSYS CFX-Solver Manager

The ANSYS CFX-Solver Manager will be launched after ANSYS CFX-Pre has closed down. 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 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. 116).

1. Ensure **Define Run** is displayed.
2. Click **Start Run**.

ANSYS 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. Click **Yes** to post-process the results when the completion message appears at the end of the run.
4. If using Standalone Mode, quit ANSYS CFX-Solver Manager.

Viewing the Results in ANSYS CFX-Post

The following topics will be discussed:

- [Creating Velocity Vector Plots](#) (p. 278)
- [Viewing Volume Fractions](#) (p. 279)
- [Displaying the Entire Airlift Reactor Geometry](#) (p. 280)

Creating Velocity Vector Plots

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, Release 11.0. You will first examine the distribution of velocities and fluid volume fraction by creating the following plots. The results will then be verified to check if the values are reasonable.

1. Right-click a blank area in the viewer and select **Predefined Camera > View Towards -Z**.
2. Zoom in as required.
3. Turn on the visibility of `SymP1`.
4. Apply the following settings to `SymP1`.

Tab	Setting	Value
Color	Mode	Variable
	Variable	Air at 25 C.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`.
7. Create a new vector named `vector 1`.
8. Apply the following settings

Tab	Setting	Value
Geometry	Definition > Locations	SymP1
	Definition > Variable	Water.Velocity
Symbol	Symbol Size	0.3

9. Click **Apply**.
10. Create a new vector plot named `vector 2`.

- Apply the following settings

Tab	Setting	Value
Geometry	Definition > Locations	SymP1
	Definition > Variable	Air at 25 C.Velocity
Symbol	Symbol Size	0.3

- Click **Apply**.
- Compare the vector fields by toggling the visibility of each and zooming in as needed.

Viewing Volume Fractions

In creating the geometry for the airlift reactor, a thin surface was used to model the draft tube. You will next plot the volume fraction of air on the thin surface.

- Right-click on a blank area in the viewer, and select **Predefined Camera > Isometric View (Y up)**.
- Zoom in as required.
- Turn off the visibility of any vector plots and turn on the visibility of `DraftTube`.
- Modify `DraftTube` by applying the following settings

Tab	Setting	Value
Color	Mode	Variable
	Variable	Air at 25 C.Volume Fraction
	Range	User Specified
	Min	0
	Max	0.02

- Click **Apply**.
 - This boundary represents one side of the thin surface. When viewing plots on thin surfaces, you must ensure that you are viewing the correct side of the thin surface.
 - The plot just created is displaying the volume fraction for air in the downcomer region of the airlift reactor. If you rotate the geometry you will see that the same plot is visible from both sides of the thin surface.
 - You will make use of the face culling feature whichs turns off the visibility of the plot on one side of the thin surface. In this case, you need to turn off the "front" faces.
- Modify `DraftTube` by applying the following settings

Tab	Setting	Value
Render	Draw Faces > Face Culling	Front Faces

- Click **Apply**.
- Rotate the image in the viewer to see the effect of face culling on `DraftTube`. You should see that the color appears only on one side: the downcomer side.
- Turn on the visibility of `DraftTube Other Side`.

- Color the `DraftTube Other Side` object using the same color settings as for `DraftTube`.

Tab	Setting	Value
Color	Mode	Variable
	Variable	Air at 25 C.Volume Fraction
	Range	User Specified
	Min	0
	Max	0.02

- Modify `DraftTube Other Side` by applying the following settings

Tab	Setting	Value
Render	Draw Faces > Face Culling	Front Faces

This will create a plot of air volume fraction on the riser side of the bubble column.

- Click **Apply**.

Rotating the geometry will now show correct plots of the air volume fraction on each side of the draft tube.

To see why face culling was needed to prevent interference between the plots on each side of the draft tube, try turning off face culling for `DraftTube` and watch the effect on the riser side (Results may vary, which is why face culling was used to prevent interference.).

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:

- Apply the following settings to `Default Transform`

Tab	Setting	Value
Definition	Instancing Info From Domain	(Cleared)
	# of Copies	12
	Apply Rotation > Axis	Y
	Apply Rotation > # of Passages	12

- Click **Apply**.

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 at 25 C.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 holdup), and the liquid velocity in the downcomer agree with the results reported by García-Calvo and Letón, for gas holdup values of 0.03 or less. At higher values of gas holdup, 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 holdup 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 (e.g., `Fluid.Superficial Velocity X`).

Many reference texts on bubble columns cite the Hughmark correlation as a standard for gas hold up and superficial gas velocity in bubble columns. However, the Hughmark correlation should not be used when liquid flow is concurrent with gas at velocities exceeding 0.1 m s⁻¹. In the airlift reactor described in this tutorial, the liquid velocity in the riser clearly exceeds 0.2 m s⁻¹ and the Hughmark correlation is therefore not applicable.

Tutorial 17: Air Conditioning Simulation

Introduction

This tutorial includes:

- [Tutorial 17 Features](#) (p. 284)
- [Overview of the Problem to Solve](#) (p. 285)
- [Defining a Simulation in ANSYS CFX-Pre](#) (p. 285)
- [Obtaining a Solution using ANSYS CFX-Solver Manager](#) (p. 295)
- [Viewing the Results in ANSYS CFX-Post](#) (p. 295)

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

- [Setting the Working Directory](#) (p. 1)
- [Changing the Display Colors](#) (p. 2)

Unless you plan on running a session file, you should copy the sample files used in this tutorial from the installation folder for your software (<CFXROOT>/examples/) to your working directory. This prevents you from overwriting source files provided with your installation. If you plan to use a session file, please refer to [Playing a Session File](#) (p. 285).

Sample files referenced by this tutorial include:

- `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.

Tutorial 17 Features

This tutorial addresses the following features of ANSYS CFX.

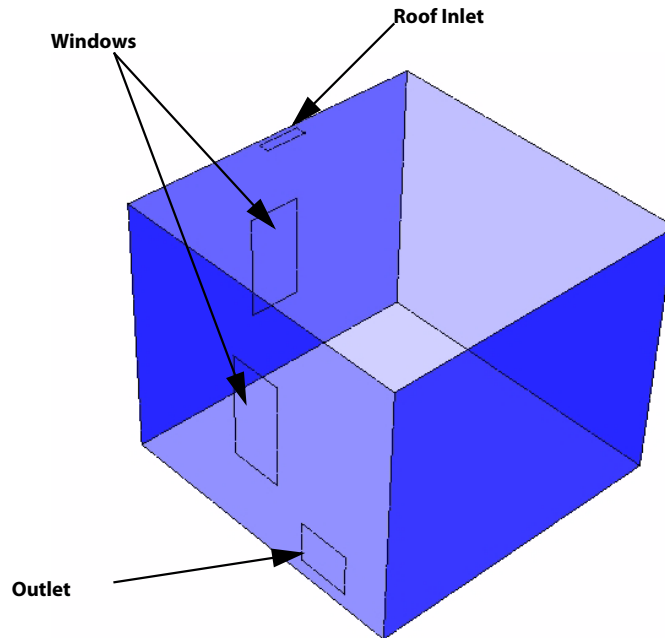
Component	Feature	Details	
ANSYS CFX-Pre	User Mode	General Mode	
	Simulation Type	Transient	
	Fluid Type	General Fluid	
	Domain Type	Single Domain	
	Turbulence Model	k-Epsilon	
	Heat Transfer	Thermal Energy	
	Radiation		
	Buoyant Flow		
	Boundary Conditions		Boundary Profile Visualization
			Inlet (Profile)
			Outlet (Subsonic)
			Wall: No-Slip
			Wall: Adiabatic
		Wall: Fixed Temperature	
	Output Control		
CEL (CFX Expression Language)			
User Fortran			
Timestep	Transient Example		
Transient Results File			
ANSYS CFX-Post	Plots	Animation	
		Isosurface	
		Point	
		Slice Plane	
	Other	Auto Annotation	
		Changing the Color Range	
		Legend	
		MPEG Generation	
		Time Step Selection	
		Title/Text	
	Transient Animation		

In this tutorial you will learn about:

- Using the Monte Carlo radiation model with a directional source of radiation.
- Setting a monitor point to observe the temperature at a prescribed location.

Overview of the Problem to Solve

This tutorial demonstrates a simple air conditioning case in a room. The room contains windows and an inlet vent for cooled air. The windows are set up to include heat and radiation sources that act to raise the temperature of the room. The inlet vent introduces cool air into the room to lower the temperature to a set level. The room also contains an outlet vent, which removes ambient air from the room.



Defining a Simulation in ANSYS CFX-Pre

This section describes the step-by-step definition of the flow physics in ANSYS CFX-Pre.

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.

Playing a Session File

If you wish to skip past these instructions, and have ANSYS CFX-Pre set up the simulation automatically, you can select **Session > Play Tutorial** from the menu in ANSYS CFX-Pre, then run the session file: `HVAC.pre`.

After performing this step, you can continue from [Obtaining a Solution using ANSYS CFX-Solver Manager](#) (p. 295).

Creating a New Simulation

1. Start ANSYS CFX-Pre.
2. Select **File > New Simulation**.
3. Select **General** and click **OK**.
4. Select **File > Save Simulation As**.
5. Under **File name**, type HVAC.
6. Click **Save**.

Importing the Mesh

1. Right-click `Mesh` and select **Import Mesh**.
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.

Creating Expressions

This tutorial requires some CEL expressions. In this tutorial, a transient simulation will be performed over 3 minutes 45 seconds with 3 second timesteps for a total of 75 timesteps. Expressions will be used to enter these values. The expressions are also used to calculate the inlet temperature of air under different conditions.

As the air conditioner will remove a specified amount of heat, the inlet vent temperature is a function of the outlet vent temperature. A CEL function is used to find the outlet temperature. A User CEL Function is used to simulate behavior of a thermostat that turns on cold air when the temperature (measured at a particular location) is above 22 °C (295.15 K) and turns off the cold air when the temperature falls below 20 °C (293.15 K).

Note: The expression for `TSensor` requires a monitor point named `Thermometer` to provide room temperature feedback to the thermostat. This will be set up later.

Importing the Expressions

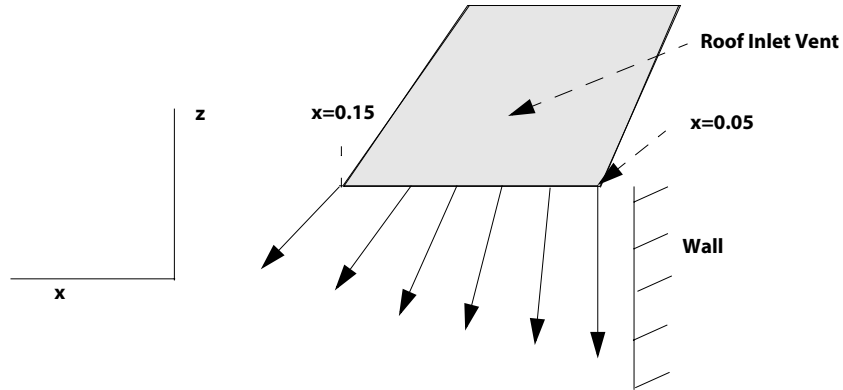
1. Select **File > Import CCL**.
2. Select the file `HVAC_expressions.ccl`.
3. Click **Open**.

The expression for `ACOn` requires a User CEL Function that indicates the thermostat output: whether the air conditioner should be on or off. This will be set up next.

Inlet Velocity Profile

Expressions are used to simulate guiding vanes at the inlet, as the following diagram shows:

Figure 1 Intended airflow direction from the roof inlet vent



The two x locations indicated on the diagram correspond to the x values across the width of the inlet vent. When x is 0.05, the z component of velocity will be -1 and the x component will be zero. When x is 0.15, the x component of velocity will be 0.5 and the z component will be -0.5. The x component of velocity varies linearly with x. The following expression can be used to calculate the x component of velocity:

$$XCompInlet = 0.5 \times \frac{x - 0.05}{0.1} = 5(x - 0.05) \quad (\text{Eqn. 1})$$

$$ZCompInlet = -1 + XCompInlet$$

Setting up the Thermostat

A Fortran subroutine that simulates the thermostat has already been written for this tutorial.

Compiling the Subroutine

You can compile the subroutine and create the required library file used by ANSYS CFX-Solver at any time before running the ANSYS 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 ANSYS CFX-Pre when creating a User CEL Function. The `cfx5mkext` command is used to compile the subroutine as described below.

Important: You must have the required Fortran compiler installed and set in your system path in order to run the `cfx5mkext` command successfully. 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.

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 semi-colon at the end of the line):

```
! system ("cfx5mkext TStat_Control.F") < 1 or die "cfx5mkext failed";
```

- This is equivalent to executing the following at an OS 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 < 1 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 (i.e., `cfx5mkext -double TStat_Control.F`) to compile the subroutine for use with double precision ANSYS CFX-Solver executables.

A subdirectory will have been created in your working directory whose name is system dependent (e.g., on IRIX it is named `irix`). This sub directory 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.

1. Close the **Command Editor** dialog box.

Creating the User CEL Function

A User CEL Function is required to link the subroutine into ANSYS CFX. The complete definition for the function is defined in two steps. First, a user routine that contains the calling name, library name, and library path is created. Then, a user function that points to the user routine, and also contains the argument and result units, is defined.

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*
	Library Name'	TStat_Control [†]
	Library Path	(Working Directory) [‡]


- *. 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.
 - †. 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.
 - ‡. Set this to your working directory.
4. Click **OK**.
 5. Create a new user function named `Thermostat Function` by selecting **Insert > Expressions, Functions and Variables > User Function** from the main menu.
 6. Apply the following settings

Tab	Setting	Value
Basic Settings	Option	User Function
	Argument Units	[K], [K], [K], []*
	Result Units	[]†

- *. These are the units for the four input arguments: `TSensor`, `TSet`, `TTol`, and `aitern`.
- †. The result will be a dimensionless integer flag of values 1 or 0.

7. Click **OK**.
 The function you have just prepared is called during the evaluation of the expression for `ACOn` (that you imported earlier). The expression is:
`Thermostat Function(TSensor, TSet, TTol, aitern)`
 It evaluates to 1 or 0, depending on whether the air conditioner should be on (1) or off (0).

Setting the Simulation Type

1. Click *Simulation Type* .
2. Apply the following settings

Tab	Setting	Value
Basic Settings	Simulation Type > Option	Transient
	Simulation Type > Time Duration > Total Time	tTotal
	Simulation Type > Time Steps > Timesteps	tStep
	Simulation Type > Initial Time > Time	0 [s]

3. Click **OK**.

Creating the Domain

1. Right click `Simulation` in the **Outline** tree view and ensure that **Automatic Default Domain** is selected. 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
General Options	Basic Settings > Location	B1.P3
	Fluids List	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 Model > Option	Monte Carlo

3. Click **OK**.

Setting Boundary Conditions

In this section you will define the locations and values of the boundary conditions.

Inlet Boundary

1. Create a new boundary condition 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
	Mass and Momentum > Mass Flow Rate	MassFlow
	Flow Direction > Option	Cartesian Components
	Flow Direction > X Component	XComplnet
	Flow Direction > Y Component	0
	Flow Direction > Z Component	ZComplnet
	Heat Transfer > Static Temperature	TIn
Plot Options	Boundary Vector	(Selected)

3. Click **OK**.

Note: Ignore the physics errors that appear. They will be fixed by setting up the rest of the simulation. The error you see is due to a reference to `Thermometer` which has not been set up yet. This will be done as part of the output control.

Outlet Boundary

1. Create a new boundary condition named `VentOut`.
2. Apply the following settings

Tab	Setting	Value
Basic Settings	Boundary Type	Outlet
	Location	VentOut
Boundary Details	Mass and Momentum > Relative Pressure	0 [Pa]

3. Click **OK**.

Window Boundary


To model incoming radiation at the window boundaries, a directional radiation source will be created. The windows will also contribute heat to the room via a fixed temperature of 26 [C].

1. Create a new boundary condition 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]

3. Apply the following settings

Tab	Setting	Value
Sources	Boundary Source	(Selected)
	Boundary Source > Sources	(Selected)

4. Create a new radiation source item by clicking *Add New Item*  and accepting the default name.
5. 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	0.33
Direction > Y Component	0.33
Direction > Z Component	-0.33

6. Apply the following setting

Tab	Setting	Value
Plot Options	Boundary Vector	(Selected)

7. Click **OK**.

The directional source of radiation is displayed.

Default Wall Boundary

The default boundary condition for any undefined surface in ANSYS CFX-Pre is a no-slip, smooth, adiabatic wall. For radiation purposes, the default wall is assumed to be a perfectly absorbing and emitting surface (emissivity = 1), and this will be preserved when setting up the boundary condition.

In this tutorial, a fixed temperature of 26 °C will be assumed to exist at the wall during the simulation. A more detailed analysis would model heat transfer through the walls, but as this tutorial is designed only for demonstration purposes, a fixed temperature wall is sufficient.


1. Modify the boundary condition named `Default Domain Default`.
2. Apply the following settings

Tab	Setting	Value
Boundary Details	Heat Transfer > Option	Temperature
	Heat Transfer > Fixed Temperature	26 [C]

3. Click **OK**.

This setting will include the `DOOR` region, which will be modeled as a wall (closed door) for simplicity. Since the region is part of the entire default boundary, it will not appear in the wireframe when the results file is opened in ANSYS CFX-Post (but can still be viewed in the **Regions** list).


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 > Relative Pressure	0 [Pa]
	Initial Conditions > Temperature > Temperature	22 [C]
	Initial Conditions > Turbulence Kinetic Energy > Fractional Intensity	(Selected)
	Initial Conditions > Turbulence Eddy Dissipation	(Selected)
	Initial Conditions > Turbulence Eddy Dissipation > Eddy Length Scale	(Selected)
	Initial Conditions > Turbulence Eddy Dissipation > Eddy Length Scale > Eddy Len. Scale	0.25 [m]
	Initial Conditions > Radiation Intensity > Blackbody Temperature	(Selected)
	Initial Conditions > Radiation Intensity > Blackbody Temperature > Blackbody Temp.	22 [C]

3. Click **OK**.

Setting Solver Control



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

3. Click **OK**.

Setting Output Control

Transient results files will be set up to record transient values of a chosen set of variables. Monitor points will be created to show the on/off status of the air conditioner, the temperature at the inlet, the temperature at the outlet, and the temperature at a prescribed thermometer location.

1. Click *Output Control* .
2. Click **Trn Results**.
3. Create a new **Transient Results** item by clicking *Add New Item*  and accept the the default name.
4. Apply the following settings to `Transient Results 1`

Setting	Value
Option	Selected Variables
Output Variables List	Pressure, Radiation Intensity, Temperature, Velocity
Output Variables Operators	(Selected)
Output Variables Operators > Output Var. Operators	All*
Output Frequency > Option	Time Interval
Output Frequency > Time Interval	tStep

*. This causes the gradients of the selected variables to be written to the transient files, along with other information.

5. Apply the following settings

Tab	Setting	Value
Monitor	Monitor Options	(Selected)

6. Create a new **Monitor Points and Expressions** item named `Temp at Inlet`.
7. Apply the following settings to `Temp at Inlet`

Setting	Value
Option	Expression
Expression Value	TIn

8. Create a new **Monitor Points and Expressions** item named `Thermometer`.
9. Apply the following settings to `Thermometer`

Setting	Value
Output Variable List	Temperature
Cartesian Coordinates	2.95, 1.5, 1.25

10. Create a new **Monitor Points and Expressions** item named `Temp at VentOut`.
11. Apply the following settings to `Temp at VentOut`


Setting	Value
Option	Expression
Expression Value	TVentOut

12. Create a new **Monitor Points and Expressions** item named `ACOnStatus`.
13. Apply the following settings to `ACOnStatus`

Setting	Value
Option	Expression
Expression Value	ACOn

14. Click **OK**.

Writing the Solver (.def) File

1. Click *Write Solver File* .
2. Apply the following settings:

Setting	Value
File name	HVAC.def
Quit CFX-Pre*	(Selected)

*. If using ANSYS CFX-Pre in Standalone Mode.

3. Ensure **Start Solver Manager** is selected and click **Save**.
4. If using Standalone Mode, quit ANSYS CFX-Pre, saving the simulation (`.cfx`) file at your discretion.

Obtaining a Solution using ANSYS CFX-Solver Manager

When ANSYS CFX-Pre has shut down and the ANSYS CFX-Solver Manager has started, obtain a solution to the CFD problem by following the instructions below.

1. Click **Start Run**.
2. When the **User Points** tab appears, click it to view the value of the temperature at `VentOut` as the solution progresses.
3. Click **Yes** to post-process the results when the completion message appears at the end of the run.
4. If using Standalone Mode, quit ANSYS CFX-Solver Manager.

Viewing the Results in ANSYS CFX-Post

The temperature of air in the house is distributed in both space and time. While the transient behavior of the temperature field can easily be shown with an animation, it is not easy to visualize a complicated 3D distribution. In order to show the key features of the temperature field, graphic objects will be produced on strategically-placed locators; Plane locators will be used to show contour plots of temperature, while Isosurfaces will be used sparingly to show the general shape of thermal plumes.


Creating Graphics Objects

Plane locators will be placed vertically through the vents and horizontally above the floor.

Plane Locators

1. Load the res file (**HVAC_001.res**) if you did not elect to load the results directly from the ANSYS 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]**, and clear **Lighting**.
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 clear **Lighting**.

Isosurface Locator

1. Click **Timestep Selector** .
The **Timestep Selector** appears.
2. Double-click the value (12s) in the **Timestep Selector**.
The Timestep is set to 12s so that the cold plume is visible.
3. Create an isosurface named **Cold Plume** which is a surface of **Temperature=19 [C]**. Use conservative values for **Temperature**.
4. Color the isosurface by **Temperature** and use the same range as for the planes. Although the color of the isosurface will not show variation (by definition), it will be consistent with the other graphic objects.
5. On the **Render** tab for the isosurface, set **Transparency** to **0.5**, and clear **Lighting**.
6. Click **Apply**.

Note: The isosurface will not be visible in some timesteps, but you will be able to see it when playing the animation (a step carried out later).

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.

1. In the tree view, double-click **Default Legend View 1**.
2. In the **Definition** tab, change **Title Mode** to **Variable**.
This will remove the locator name from the legend.
3. Click the **Appearance** tab, then:
 - a. Change **Precision** to **2, Fixed**.
 - b. Change **Text Height** to **0.03**.
4. Click **Apply**.

A label will be used to show the simulation time and the temperature of the thermometer which controls the thermostat. This will be especially useful for the animation which is created later in this tutorial.

Before creating the label, you will need to support the expression for T_{Sensor} by creating a point called Thermometer at the location of the sensor thermometer. This point will replace the monitor point called Thermometer which was used during the solver run, but no longer exists.


Note: The actual thermometer data generated during the run was stored in the results file, but is not easily accessible, and cannot currently be used in an auto-annotation label.

Creating a Point for the Thermometer

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**.



Now the expression T_{Sensor} will once again measure temperature at the prescribed location.


Creating the Text Label

1. Click **Text** .
2. Accept the default name and click **OK**.
3. Set **Text String** to Time Elapsed
4. Select **Embed Auto Annotation**.
The full text string should now be $\text{Time Elapsed: <aa>}$. The <aa> represents the location where the auto annotation will be substituted.
5. Set **Type** to Time Value .
This will show the amount of simulated time that has passed in the simulation.
6. Click **More**.
This adds a second line of text to the text object.
7. Set **Text String** to $\text{Sensor Temperature:}$
8. Select **Embed Auto Annotation**.
9. Set **Type** to Expression .
10. Set **Expression** to T_{Sensor} .
11. Click the **Appearance** tab, change **Height** to 0.03 , then click **Apply**.

Ensure the visibility check box next to Text 1 is selected. A label appears at the top of the figure. The large font is used so that the text will be clearly visible in the animation which will be produced in the next section.


Creating an Animation


1. Ensure that the view is set to **Isometric View (Z up)**.
2. Click **Timestep Selector** .
The **Timestep Selector** appears.
3. Double-click the first time value (0 s) in the **Timestep Selector**.
4. Click **Animation**  found in the toolbar.
The **Animation** dialog box appears.
5. In the **Animation** dialog box:




- a. Click *New*  to create *KeyframeNo1*.
- b. Highlight *KeyframeNo1*, change **# 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 8.8 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.

6. Load the last time value (225 s) using the **Timestep Selector** dialog box.
7. In the **Animation** dialog box:
 - a. Click *New*  to create *KeyframeNo2*.

The **# of Frames** parameter has no effect for the last keyframe, so leave it at the default value.
 - b. Click *More Animation Options*  to expand the **Animation** dialog box.
 - c. Select **Save MPEG**.
 - d. Specify a file name for the MPEG file.
 - e. Click the **Options** button.
 - f. Change **MPEG Size** to 720 × 480 (or a similar resolution).
 - g. Click the **Advanced** tab, and note the **Quality** setting.

If your MPEG player does not play the MPEG, you can try using the **Low** or **Custom** quality settings.
 - h. Click **OK**.
 - i. Click *To Beginning*  to rewind the active key frame to *KeyframeNo1*.
 - j. 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).
8. Click *Play the animation* .
9. If prompted to overwrite an existing movie, click **Overwrite**.

The animation plays and builds an **.mpg** file.
10. When you have finished, quit ANSYS CFX-Post.

Further Steps

1. This tutorial uses an aggressive value for the flow rate of air, a coarse mesh, and the timesteps are too large for a satisfactory analysis. Running this tutorial with a finer mesh, a flow rate of air that is closer to 5 changes of air per hour ($0.03 \text{ m}^3 \text{ s}^{-1}$), and smaller timesteps will produce more accurate results.
2. Running the simulation for a longer total time period will allow you to see more on/off cycles of the thermostat.

Tutorial 18: Combustion and Radiation in a Can Combustor

Introduction

This tutorial includes:

- [Tutorial 18 Features](#) (p. 300)
- [Overview of the Problem to Solve](#) (p. 301)
- [Using Eddy Dissipation and P1 Models](#) (p. 301)
- [Defining a Simulation in ANSYS CFX-Pre](#) (p. 302)
- [Obtaining a Solution using ANSYS CFX-Solver Manager](#) (p. 307)
- [Viewing the Results in ANSYS CFX-Post](#) (p. 308)
- [Laminar Flamelet and Discrete Transfer Models](#) (p. 311)
- [Further Postprocessing](#) (p. 316)

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

- [Setting the Working Directory](#) (p. 1)
- [Changing the Display Colors](#) (p. 2)

Unless you plan on running a session file, you should copy the sample files used in this tutorial from the installation folder for your software (<CFXROOT>/**examples**/) to your working directory. This prevents you from overwriting source files provided with your installation. If you plan to use a session file, please refer to [Playing a Session File](#) (p. 302).

Sample files referenced by this tutorial include:

- **CombustorMesh.gtm**
- **CombustorEDM.pre**
- **CombustorFlamelet.pre**
- **CombustorEDM.cfx**

Tutorial 18 Features

This tutorial addresses the following features of ANSYS CFX.

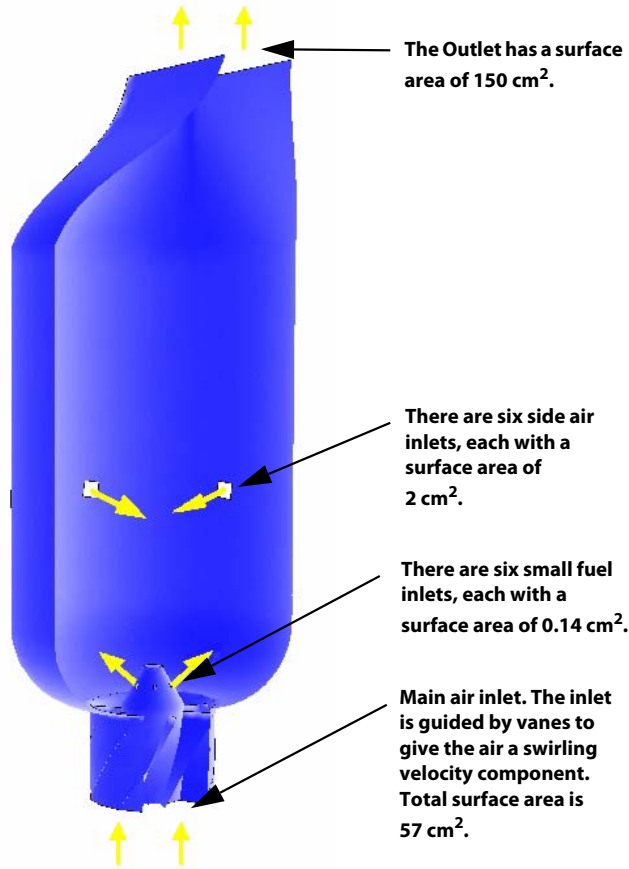
Component	Feature	Details	
ANSYS CFX-Pre	User Mode	General Mode	
	Simulation Type	Steady State	
	Fluid Type	Reacting Mixture	
	Domain Type	Single Domain	
	Turbulence Model	k-Epsilon	
	Heat Transfer	Thermal Energy	
	Combustion		
	Radiation		
	Boundary Conditions	Inlet (Subsonic)	
		Outlet (Subsonic)	
		Wall: No-Slip	
Wall: Adiabatic			
	Wall: Thin Surface		
Timestep	Physical Time Scale		
ANSYS CFX-Post	Plots	Outline Plot (Wireframe)	
		Sampling Plane	
		Slice Plane	
		Vector	
	Other	Changing the Color Range	
		Color map	
	Legend		
	Quantitative Calculation		

In this tutorial you will learn about:

- Creating thin surfaces for the inlet vanes.
- Using a Reacting Mixture.
- Using the Eddy Dissipation Combustion Model.
- Using the Flamelet Model.
- Changing the Combustion model in a simulation.
- Using the P1 Radiation Model in ANSYS CFX-Pre.
- Using the Discrete Transfer Radiation Model in ANSYS CFX-Pre.
- Using the NOx model in ANSYS CFX-Pre.
- Changing object color maps in ANSYS CFX-Post to prepare a greyscale image.

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. The basic geometry is shown below with a section of the outer wall cut away.



Using Eddy Dissipation and P1 Models

This tutorial demonstrates two different combustion and radiation model combinations. The first uses the Eddy Dissipation Combustion model with the P1 Radiation model; the NOx model is also included. The second uses the Laminar Flamelet model with the Discrete Transfer Radiation model. If you wish to use the Flamelet Combustion model and Discrete Transfer Radiation model, see [Laminar Flamelet and Discrete Transfer Models](#) (p. 311), otherwise continue from this point.

Defining a Simulation in ANSYS CFX-Pre

You will define a domain that includes a variable composition mixture. These mixtures are used to model combusting and reacting flows in ANSYS CFX.

Playing a Session File

If you wish to skip past these instructions, and have ANSYS CFX-Pre set up the simulation automatically, you can select **Session > Play Tutorial** from the menu in ANSYS CFX-Pre, then run the session file: `CombustorEDM.pre`. After you have played the session file as described in earlier tutorials under [Playing the Session File and Starting ANSYS CFX-Solver Manager](#) (p. 87), proceed to [Obtaining a Solution using ANSYS CFX-Solver Manager](#) (p. 307).

Creating a New Simulation

1. Start ANSYS CFX-Pre.
2. Select **File > New Simulation**.
3. Select **General** and click **OK**.
4. Select **File > Save Simulation As**.
5. Under **File name**, type `CombustorEDM`.
6. Click **Save**.

Importing the Mesh

1. Right-click `Mesh` and select **Import Mesh**.
2. Apply the following setting

Setting	Value
File name	CombustorMesh.gtm

3. Click **Open**.

Creating a Reacting Mixture

To allow combustion modeling, you must create a variable composition mixture.

To create the variable composition mixture

1. Create a new material named `Methane Air Mixture`.
2. Apply the following settings

Tab	Setting	Value
Basic Settings	Option	Reacting Mixture
	Material Group	Gas Phase Combustion
	Reactions List	Methane Air WD1 NO PDF*

Tab	Setting	Value
Mixture Properties	Mixture Properties	(Selected)
	Mixture Properties > Radiation Properties > Refractive Index	(Selected) [†]
	Mixture Properties > Radiation Properties > Absorption Coefficient	(Selected)
	Mixture Properties > Radiation Properties > Scattering Coefficient	(Selected)

*. 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.

†. Setting the radiation properties explicitly will significantly shorten the solution time since the ANSYS CFX-Solver will not have to calculate radiation mixture properties.

3. Click **OK**.

Creating the Domain

1. Right click *Simulation* in the **Outline** tree view and ensure that **Automatic Default Domain** is selected. 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
General Options	Basic Settings > Locations	B152, B153, B154, B155, B156
	Basic Settings > Fluids List	Methane Air Mixture
	Domain Models > Pressure > Reference Pressure	1 [atm] [*]
Fluid Models	Heat Transfer > Option	Thermal Energy
	Reaction or Combustion > Option	Eddy Dissipation
	Reaction or Combustion > Eddy Dissipation Model Coefficient B	(Selected)
	Reaction or Combustion > Eddy Dissipation Model Coefficient B > EDM Coeff. B	0.5 [†]
	Thermal Radiation Model > Option	P 1
	Component Details > N2	(Selected)
	Component Details > N2 > Option	Constraint

*. It is important to set a realistic reference pressure in this tutorial because the components of *Methane Air Mixture* are ideal gases.

†. 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.

3. Click **OK**.

Creating the Boundary Conditions

Fuel Inlet Boundary

1. Create a new boundary condition named `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
	Component Details > CH4 > Mass Fraction	1

3. Click **OK**.

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 fuel inlet to give the incoming air a swirling velocity.

1. Create a new boundary condition 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
	Component Details > O2 > Mass Fraction	0.232*

*. The remaining mass fraction at the inlet will be made up from the constraint component, `N2`.

3. Click **OK**.

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 condition named `secairin`.
2. Apply the following settings

Tab	Setting	Value
Basic Settings	Boundary Type	Inlet
	Location	secairin

Tab	Setting	Value
Boundary Details	Mass and Momentum > Option	Normal Speed
	Mass and Momentum > Normal Speed	6 [m s ⁻¹]
	Heat Transfer > Option	Static Temperature
	Heat Transfer > Static Temperature	300 [K]
	Component Details	O2
	Component Details > O2 > Mass Fraction	0.232*

*. The remaining mass fraction at the inlet will be made up from the constraint component, N2.

3. Click **OK**.

Outlet Boundary

1. Create a new boundary condition 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**.

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 ANSYS CFX-Pre, you must specify a wall boundary condition on each side of the vanes. The **Create Thin Surface Partner** feature in ANSYS CFX-Pre will automatically match the other side of a thin surface if you pick just a single side.

You will first create a new region which contains one side of each of the eight vanes, then use the **Create Thin Surface Partner** feature to match the other side.

1. Create a new composite region named `Vane Surfaces`.
2. Apply the following settings

Tab	Setting	Value
Basic Settings	Dimension (Filter)	2D*
	Region List	F129.152, F132.152, F136.152, F138.152, F141.152, F145.152, F147.152, F150.152

*. This will filter out the 3D regions, leaving only 2D regions

3. Click **OK**.
4. Create a new boundary condition named `vanes`.
5. Apply the following settings

Tab	Setting	Value
Basic Settings	Boundary Type	Wall
	Location	Vane Surfaces
	Create Thin Surface Partner	(Selected)*

*. This feature will attempt to match all primitives specified in the location list to create a thin surface boundary condition.

6. Click **OK**.


Default Wall Boundary

The default boundary condition for any undefined surface in ANSYS 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.


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	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 > Turbulence Eddy Dissipation	(Selected)
	Initial Conditions > Turbulence Eddy Dissipation > Option	Automatic
	Initial Conditions > Component Details	O2
	Initial Conditions > Component Details > O2> Option	Automatic with Value
	Initial Conditions > Component Details > O2 > Mass Fraction	0.232*
	Initial Conditions > Component Details	CO2
	Initial Conditions > Component Details > CO2 > Option	Automatic with Value
	Initial Conditions > Component Details > CO2 > Mass Fraction	0.01
	Initial Conditions > Component Details	H2O
	Initial Conditions > Component Details> H2O > Option	Automatic with Value
	Initial Conditions > Component Details > H2O > Mass Fraction	0.01

- *. 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 (CO₂ and H₂O) is needed for the EDM model to initiate combustion.
3. Click **OK**.


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**.

Writing the Solver (.def) File

1. Click *Write Solver File* .
2. Apply the following settings:

Setting	Value
File name	CombustorEDM.def
Quit CFX-Pre*	(Selected)

*. If using ANSYS CFX-Pre in Standalone Mode.

3. Ensure **Start Solver Manager** is selected and click **Save**.
4. If using Standalone Mode, quit ANSYS CFX-Pre, saving the simulation (.cfx) file.

Obtaining a Solution using ANSYS CFX-Solver Manager

The ANSYS CFX-Solver Manager will be launched after ANSYS CFX-Pre saves the definition 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. 116).

1. Ensure **Define Run** is displayed.

Definition File should be set to **CombustorEDM.def**.

2. Click **Start Run**.
ANSYS 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. Click **Yes** to post-process the results.
4. If using Standalone Mode, quit ANSYS CFX-Solver Manager.

Viewing the Results in ANSYS CFX-Post

When ANSYS CFX-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.

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
Color	Mode	Variable
	Mode > Variable	Temperature

3. Click **Apply**.

The large area of high temperature through most of the vessel is due to forced convection.

Note: Later in this tutorial (see [Laminar Flamelet and Discrete Transfer Models](#) (p. 311)), 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.

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**.

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**.

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 **Tools** 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.

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 named `Vector 1`.
4. Apply the following settings

Tab	Setting	Value
Geometry	Definition > Locations	Plane 1
Symbol	Symbol Size	2

5. Click **Apply**.
6. Create a new plane named `Plane 2`.
7. Apply the following settings

Tab	Setting	Value
Geometry	Definition > Method	XY Plane
	Definition > Z	0.03
	Plane Bounds > Type	Rectangular
	Plane Bounds > X Size	0.5 [m]
	Plane Bounds > Y Size	0.5 [m]
	Plane Type > Sample	(Selected)
	Plane Type > X Samples	30
	Plane Type > Y Samples	30
Render	Draw Faces	(Cleared)

8. Click **Apply**.
9. Modify `Vector 1`.
10. Apply the following setting

Tab	Setting	Value
Geometry	Definition > Locations	Plane 2

11. Click **Apply**.

To view the swirling velocity field, right-click in the viewer and select **Predefined Camera > View Towards -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.

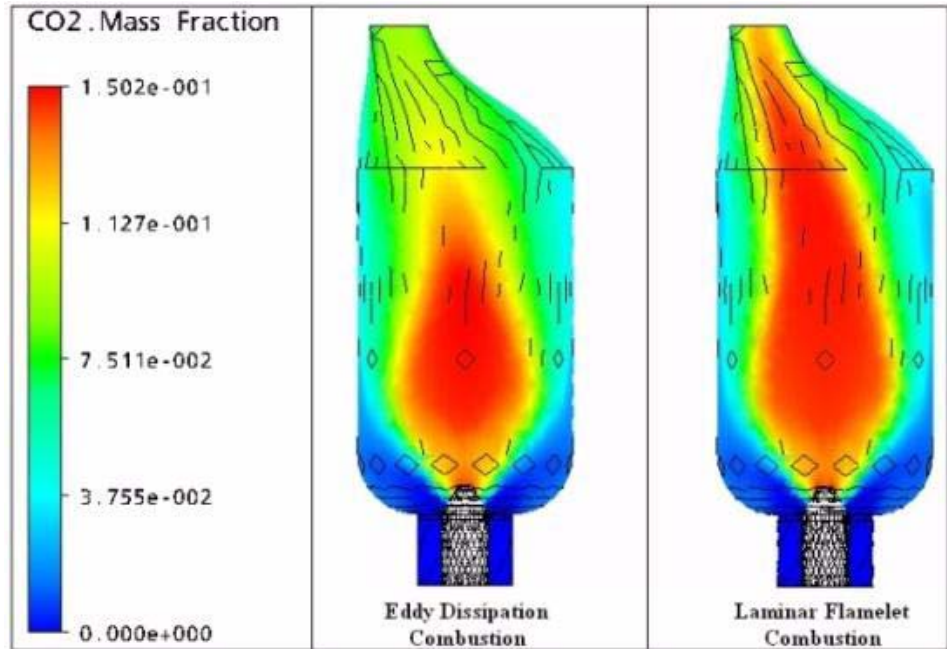
Viewing Radiation

Try examining the distribution of `Incident Radiation` and `Radiation Intensity` throughout the domain.

When you are finished, quit ANSYS CFX-Post.

Laminar Flamelet and Discrete Transfer Models

In this second part of the tutorial, you will start with the simulation from the first part of the tutorial and modify it to use the Laminar Flamelet combustion and Discrete Transfer radiation models. 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.



Playing a Session File

If you wish to skip past these instructions, and have ANSYS CFX-Pre set up the simulation automatically, you can select **Session > Play Tutorial** from the menu in ANSYS CFX-Pre, then run the session file: `combustorFlamelet.pre`. After you have played the session file as described in earlier tutorials under [Playing the Session File and Starting ANSYS CFX-Solver Manager](#) (p. 87), proceed to [Obtaining a Solution](#) (p. 314).

Creating a New Simulation

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 ANSYS CFX-Pre and then play the session file `combustorEDM.pre`. The simulation file `combustorEDM.cfx` will be created.
2. Start ANSYS CFX-Pre (unless it is already running).
3. Select **File > Open Simulation**.
4. Load the simulation named `combustorEDM.cfx`.
The simulation from the first part of this tutorial is loaded.
5. Select **File > Save Simulation As**.

- 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.

Modifying the Reacting Mixture

A flamelet library will be used to create the variable composition mixture.

- Expand **Materials** and open **Methane Air Mixture** for editing.
- Apply the following settings

Tab	Setting	Value
Basic Settings	Reactions List	Methane Air FLL STP and NO PDF*

*. Click  to display the **Reactions List** dialog box, then click *Import Library Data*



and select the appropriate reaction to import.

- Click **OK**.

Note: Some physics validation messages appear after this reaction is selected. In this situation, the messages can be safely ignored as the physics will be corrected once the domains and boundary conditions are modified.

Modifying the Domain

- Double-click the **Default Domain**.
- Apply the following settings

Tab	Setting	Value
Fluid Models	Reaction or Combustion > Option	PDF Flamelet
	Thermal Radiation Model > Option	Discrete Transfer
	Component Details	N2
	Component Details > N2 > Option	Constraint
	Component Details	NO
	Component Details > NO > Option	Transport Equation
	Component Details	(All other components)*
	Component Details > (All other components) > Option	Automatic

*. Select these one at a time and check each of them.

- Click **OK**.

Modifying the Boundary Conditions

Fuel Inlet Boundary

- Modify the boundary condition named **fuelin**.
- 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

3. Click **OK**.

Bottom Air Inlet Boundary

1. Modify the boundary condition named `airin`.
2. Apply the following settings

Tab	Setting	Value
Boundary Details	Mixture > Option	Oxidiser
	Component Details	NO
	Component Details > NO > Option	Mass Fraction
	Component Details > NO > Mass Fraction	0

3. Click **OK**.


Side Air Inlet Boundary

1. Modify the boundary condition named `secairin`.
2. Apply the following settings

Tab	Setting	Value
Boundary Details	Mixture > Option	Oxidiser
	Component Details	NO
	Component Details > NO > Option	Mass Fraction
	Component Details > NO > Mass Fraction	0

3. Click **OK**.

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

3. Click **OK**.

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**.

Writing the Solver (.def) File

1. Click *Write Solver File* .
2. Apply the following settings:

Setting	Value
File name	CombustorFlamelet.def
Quit CFX-Pre *	(Selected)

*. If using ANSYS CFX-Pre in Standalone Mode.

3. Ensure **Start Solver Manager** is selected and click **Save**.
4. If using Standalone Mode, quit ANSYS CFX-Pre, saving the simulation (.cfx) file at your discretion.

Obtaining a Solution

When ANSYS CFX-Pre has shut down and the ANSYS CFX-Solver Manager has started, obtain a solution to the CFD problem by following the instructions below.

1. Ensure **Define Run** is displayed.
Definition File should be set to `CombustorFlamelet.def`.
2. Click **Start Run**.
ANSYS 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. Click **Yes** to post-process the results.
4. If using Standalone Mode, quit ANSYS CFX-Solver Manager.

Viewing the Results

Viewing Temperature within the Domain

1. Create a new plane named `Plane 1`.

Note: If ANSYS CFX-Post was not closed since `CombustorEDM.def` was processed, all meshes and locators from that session will be retained and updated when the `CombustorFlamelet.def` is opened. In this way `Plane 1` does not need to be remade.

2. Apply the following settings

Tab	Setting	Value
Geometry	Definition > Method	ZX Plane
	Definition > Y	0
Color	Mode	Variable
	Mode > Variable	Temperature

3. Click **Apply**.

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**.

Calculating NO Concentration

The next calculation shows the amount of NO at the outlet.

1. Select **Tools > Function Calculator** or click the **Tools** 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**.

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**.

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 **Tools** 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 0.4% CO by mass in the outlet stream.

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 **Add to current results** 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.

Tutorial 19: Cavitation Around a Hydrofoil

Introduction

This tutorial includes:

- [Tutorial 19 Features](#) (p. 318)
- [Overview of the Problem to Solve](#) (p. 319)
- [Creating an Initial Simulation](#) (p. 319)
- [Obtaining an Initial Solution using ANSYS CFX-Solver Manager](#) (p. 323)
- [Viewing the Results of the Initial Simulation](#) (p. 324)
- [Preparing a Simulation with Cavitation](#) (p. 326)
- [Obtaining a Cavitation Solution using ANSYS CFX-Solver Manager](#) (p. 328)
- [Viewing the Results of the Cavitation Simulation](#) (p. 328)

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

- [Setting the Working Directory](#) (p. 1)
- [Changing the Display Colors](#) (p. 2)

Unless you plan on running a session file, you should copy the sample files used in this tutorial from the installation folder for your software (`<CFXROOT>/examples/`) to your working directory. This prevents you from overwriting source files provided with your installation. If you plan to use a session file, please refer to [Playing a Session File](#) (p. 319).

Sample files referenced by this tutorial include:

- **HydrofoilExperimentalCp.csv**
- **HydrofoilGrid.def**
- **HydrofoilIni.pre**
- **Hydrofoil.pre**
- **HydrofoilIni_001.res**

Tutorial 19 Features

This tutorial addresses the following features of ANSYS CFX.

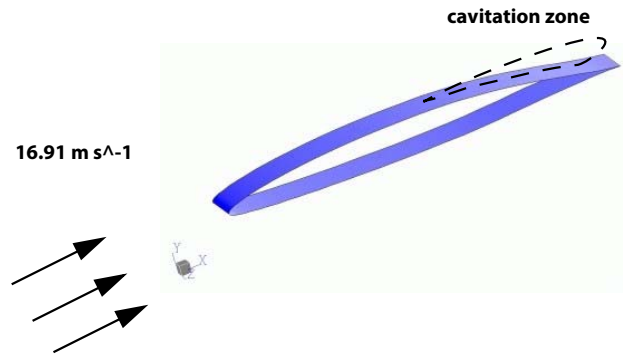
Component	Feature	Details	
ANSYS CFX-Pre	User Mode	General Mode	
	Simulation Type	Steady State	
	Fluid Type	General Fluid	
	Domain Type	Single Domain	
	Turbulence Model	k-Epsilon	
	Heat Transfer	Isothermal	
	Multiphase		
	Boundary Conditions		Inlet (Subsonic)
			Outlet (Subsonic)
			Symmetry Plane
		Wall: No-Slip	
		Wall: Free-Slip	
	Timestep	Physical Time Scale	
ANSYS CFX-Solver Manager	Restart		
ANSYS CFX-Post	Plots	Contour	
		Line Locator	
		Polyline	
		Slice Plane	
		Streamline	
		Vector	
	Other		Chart Creation
			Data Export
			Printing
			Title/Text
		Variable Details View	

In this tutorial you will learn about:

- Modeling flow with cavitation.
- Using vector reduction in ANSYS CFX-Post to clarify a vector plot with many arrows.
- Importing and exporting data along a polyline.
- Plotting computed and experimental results.

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.

Creating an Initial Simulation

This section describes the step-by-step definition of the flow physics in ANSYS CFX-Pre.

Playing a Session File

If you wish to skip past these instructions, and have ANSYS CFX-Pre set up the simulation automatically, you can select **Session > Play Tutorial** from the menu in ANSYS CFX-Pre, then run the session file: **HydrofoilIni.pre**. After you have played the session file as described in earlier tutorials under [Playing the Session File and Starting ANSYS CFX-Solver Manager](#) (p. 87), proceed to [Obtaining an Initial Solution using ANSYS CFX-Solver Manager](#) (p. 323).

Defining the Simulation

1. Start ANSYS CFX-Pre.
2. Select **File > New Simulation**.
3. Select **General** and click **OK**.
4. Select **File > Save Simulation As**.
5. Under **File name**, type `HydrofoilIni`.
6. Click **Save**.

Importing the Mesh

1. Right-click `Mesh` and select **Import Mesh**. The **Import Mesh** dialog box appears.
2. Apply the following settings

Setting	Value
File type	CFX-Solver (*.def, *.ref, *.trn, *.bak)
File name	HydrofoilGrid.def

- Click **Open**.
- Right-click a blank area in the viewer and select **Predefined Camera > View Towards -Z**.

Loading Materials

Since this tutorial uses `Water Vapour at 25 C` and `Water at 25 C` you need to load these materials.

- In the **Outline** tree view, right-click `Materials` and select **Import Library Data**. The **Select Library Data to Import** dialog box is displayed.
- Expand **Water Data**.
- Select both `Water Vapour at 25 C` and `Water at 25 C` by holding <Ctrl> when selecting.
- Click **OK**.

Creating the Domain

The fluid domain used for this simulation contains liquid water and water vapour. The volume fractions are initially set so that the domain is filled entirely with liquid.

- Right click `Simulation` in the **Outline** tree view and ensure that **Automatic Default Domain** is selected. A domain named `Default Domain` should now appear under the `Simulation` branch.
- Double click `Default Domain` and apply the following settings

Tab	Setting	Value
General Options	Basic Settings > Fluids List*	Water at 25 C, Water Vapour at 25 C
	Domain Models > Pressure > Reference Pressure	0 [atm]
Fluid Models	Multiphase Options > Homogeneous Model	(Selected)
	Heat Transfer > Option	Isothermal
	Heat Transfer > Fluid Temperature	300 [K]
	Turbulence > Option	k-Epsilon

*. These two fluids have consistent reference enthalpies.

- Click **OK**.

Creating the Boundary Conditions

The simulation requires inlet, outlet, wall and symmetry plane boundary conditions. The regions for these boundary conditions were imported with the grid file.

- Inlet Boundary**
1. Create a new boundary condition 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 > Value	0.03
	Turbulence > Eddy Len. Scale	0.0076 [m]
Fluid Values	Boundary Conditions	Water at 25 C
	Boundary Conditions > Water at 25 C > Volume Fraction > Volume Fraction	1
	Boundary Conditions	Water Vapour at 25 C
	Boundary Conditions > Water Vapour at 25 C > Volume Fraction > Volume Fraction	0

3. Click **OK**.

- Outlet Boundary**
1. Create a new boundary condition 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**.

- Free Slip Wall Boundary**
1. Create a new boundary condition named `SlipWalls`.
 2. Apply the following settings

Tab	Setting	Value
Basic Settings	Boundary Type	Wall
	Location	BOT, TOP
Boundary Details	Wall Influence on Flow > Option	Free Slip

3. Click **OK**.

- Symmetry Plane Boundaries**
1. Create a new boundary condition 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 condition named *Sym2*.
2. Apply the following settings

Tab	Setting	Value
Basic Settings	Boundary Type	Symmetry
	Location	SYM2

3. Click **OK**.


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]
	Initial Conditions > Turbulence Eddy Dissipation	(Selected)
Fluid Settings	Fluid Specific Initialization	Water at 25 C
	Fluid Specific Initialization > Water at 25 C	(Selected)
	Fluid Specific Initialization > Water at 25 C > Initial Conditions > Volume Fraction > Option	Automatic with Value
	Fluid Specific Initialization > Water at 25 C > Initial Conditions > Volume Fraction > Volume Fraction	1
	Fluid Specific Initialization	Water Vapour at 25 C
	Fluid Specific Initialization > Water Vapour at 25 C	(Selected)
	Fluid Specific Initialization > Water Vapour at 25 C > Initial Conditions > Volume Fraction > Option	Automatic with Value
	Fluid Specific Initialization > Water Vapour at 25 C > Initial Conditions > Volume Fraction > Volume Fraction	0

3. Click **OK**.

Setting Solver Control


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

Tab	Setting	Value
Basic Settings	Convergence Control > Max. Iterations	35
	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**.

Writing the Solver (.def) File

1. Click *Write Solver File* .
2. Apply the following settings

Setting	Value
File name	Hydrofoillni.def
Quit CFX-Pre*	(Selected)

*. If using ANSYS CFX-Pre in Standalone Mode.

3. Ensure **Start Solver Manager** is selected and click **Save**.
4. Quit ANSYS CFX-Pre, saving the simulation (.cfx) file at your discretion.

Obtaining an Initial Solution using ANSYS 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**.
ANSYS 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. Click **Yes** to post-process the results.
4. If using Standalone Mode, quit ANSYS CFX-Solver Manager.

Viewing the Results of the Initial Simulation

The following topics will be discussed:

- [Plotting Pressure Distribution Data](#) (p. 324)
- [Exporting Pressure Distribution Data](#) (p. 325)
- [Saving the Post-Processing State](#) (p. 326)

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 Towards -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	Draw 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. Create a new variable named `Pressure Coefficient`.
9. Apply the following settings

Setting	Value
Expression	$(\text{Pressure}-51957[\text{Pa}]) / (0.5 * 996.2[\text{kg m}^{-3}] * 16.91[\text{m s}^{-1}]^2)$

10. Click **Apply**.
11. Create a new variable named `Chord`.
12. Apply the following settings

Setting	Value
Expression	(X-minVal(X)@Foil)/(maxVal(X)@Foil-minVal(X)@Foil)

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.

13. 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
Chart	Title	Pressure Coefficient Distribution
	Labels > Use Data For Axis Label	(Cleared)
	Labels > X Axis	Normalized Chord Position
	Labels > Y Axis	Pressure Coefficient
Chart Line 1	Line Name	Solver Cp
	Location	Foil
	X Axis > Variable	Chord
	Y Axis > Variable	Pressure Coefficient
Axes	X Axis > Determine Ranges Automatically	(Cleared)
	X Axis > Min	0
	X Axis > Max	1
	Y Axis > Determine Ranges Automatically	(Cleared)
	Y Axis > Min	-0.5
	Y Axis > Max	0.4
	Y Axis > Invert Axis	(Selected)

4. Click **Apply**.
5. The chart appears on the **Chart Viewer** tab.

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**. The **Export** dialog box appears
2. Apply the following settings

Tab	Setting	Value
Options	File	NoCavCpData.csv
	Locations	Foil
	Export Geometry Information	(Selected)*
	Select Variables	Chord, Pressure Coefficient

*. 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.

Saving the Post-Processing State

You will need to save the post-processing state for use later in this tutorial.

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.

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.

Playing a Session File

This section describes the step-by-step definition of the flow physics in ANSYS CFX-Pre. If you wish to skip past these instructions, and have ANSYS CFX-Pre set up the simulation automatically, you can select **Session > Play Tutorial** from the menu in ANSYS CFX-Pre, then run the session file: **Hydrofoil1.pre**. After you have played the session file as described in earlier tutorials under [Playing the Session File and Starting ANSYS CFX-Solver Manager](#) (p. 87), proceed to [Obtaining a Cavitation Solution using ANSYS CFX-Solver Manager](#) (p. 328).

Modifying the Initial Simulation

1. Start ANSYS CFX-Pre.
2. Select **File > Open Simulation**.
3. Select **Hydrofoil1Ini_001.res** and click **Open**.
4. Save the simulation as **Hydrofoil1.cfx**.

Adding Cavitation


1. Double-click `Default Domain` in the **Outline** tree view.
2. Apply the following settings

Tab	Setting	Value
Fluid Pairs	Fluid Pairs > Water at 25 C Water Vapour at 25 C > Mass Transfer > Option	Cavitation
	Fluid Pairs > Water at 25 C Water Vapour at 25 C > Mass Transfer > Cavitation > Saturation Pressure	(Selected)
	Fluid Pairs > Water at 25 C Water Vapour at 25 C > Mass Transfer > Cavitation > Saturation Pressure > Saturation Pressure	3574 [Pa] [*]

*. 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**.

Modifying Solver Control


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

Tab	Setting	Value
Basic Settings	Convergence Control > Max. Iterations	150 [*]

*. This allows up to 150 further iterations.

3. Click **OK**.

Writing the Solver (.def) File

1. Click *Write Solver File* .
2. Apply the following settings:

Setting	Value
File name	Hydrofoil.def
Quit CFX-Pre [*]	(Selected)

*. If using ANSYS CFX-Pre in Standalone Mode.

3. Ensure **Start Solver Manager** is selected and click **Save**.
4. If using Standalone Mode, quit ANSYS CFX-Pre, saving the simulation (`.cfx`) file at your discretion.

Obtaining a Cavitation Solution using ANSYS CFX-Solver Manager

1. Ensure the **Define Run** dialog box is displayed.
Definition File should be set to `Hydrofoil.def`.
2. Set **Initial Values File** to `HydrofoilIni_001.res`.
This is the solution from the starting-point run.
3. Click **Start Run**.
4. You may see a notice that the mesh from the initial values file will be used. This mesh is the same as in the definition file. Click **OK** to continue.
ANSYS 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.
5. Click **Yes** to post-process the results.
6. If using Standalone Mode, quit ANSYS CFX-Solver Manager.

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, the second results file will be loaded into the initial state file, so you will not need to load the initial state file and you may skip the first few steps related to loading `Cp_plot.cst`.

1. Select **File > Load State**.
2. Clear **Load results**.
3. Select `Cp_plot.cst`.
4. Click **Open**.
5. Click the **Chart Viewer** tab.
6. In the **Outline** workspace, select `Report` and double-click `Pressure Coefficient Distribution`.
7. Click the **Chart Line 1** tab.
8. Apply the following setting

Tab	Setting	Value
Chart Line 1	Line 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** to update `Chart Line 1`.
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. In the **Outline** workspace, select `Report` and double-click `Pressure Coefficient Distribution`.
14. Click the **Chart Line 1** tab.
15. Click **New Line**.
A new tab named **Chart Line 2** replaces the **Chart Line 1** tab.
16. Apply the following settings

Tab	Setting	Value
Chart Line 2	Line Name	Solver Cp - no cavitation
	Location	NoCavCpPolyline
	X Axis > Variable	Chord on NoCavCpPolyline
	Y Axis > Variable	Pressure Coefficient on NoCavCpPolyline

17. 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.
18. Click **New Line**.
19. Apply the following settings

Tab	Setting	Value
Chart Line 3	Type	From File
	Line Name	Experimental Cp - with cavitation
	File	HydrofoilExperimentalCp.csv
	Appearance > Line Style	None

Tab	Setting	Value
	Appearance > Symbols	Rectangle

20. Click **Apply**.

The chart line (containing experimental data) is created, added to the chart object, and displayed on the **Chart Viewer** tab.

21. If you wish to print the chart, select **File > Print** from the main menu while the **Chart Viewer** tab is selected. This will allow you to print the chart to an image file.

22. When you are finished, close ANSYS CFX-Post.

Tutorial 20: Fluid Structure Interaction and Mesh Deformation

Introduction

This tutorial includes:

- [Tutorial 20 Features](#) (p. 332)
- [Overview of the Problem to Solve](#) (p. 333)
- [Using CEL Expressions to Govern Mesh Deformation](#) (p. 334)
- [Using a Junction Box Routine to Govern Mesh Deformation](#) (p. 343)

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

- [Setting the Working Directory](#) (p. 1)
- [Changing the Display Colors](#) (p. 2)

Unless you plan on running a session file, you should copy the sample files used in this tutorial from the installation folder for your software (<CFXR00T>/**examples**/) to your working directory. This prevents you from overwriting source files provided with your installation. If you plan to use a session file, please refer to [Playing a Session File](#) (p. 334).

Sample files referenced by this tutorial include:

- **valvefsi**
Copy the contents of this directory (but not the directory itself) into your working directory.
- **ValveFSI.pre**
- **ValveFSI_expressions.ccl**
- **ValveFSIUserF.pre**
- **ValveFSI.out**

Tutorial 20 Features

This tutorial addresses the following features of ANSYS CFX.

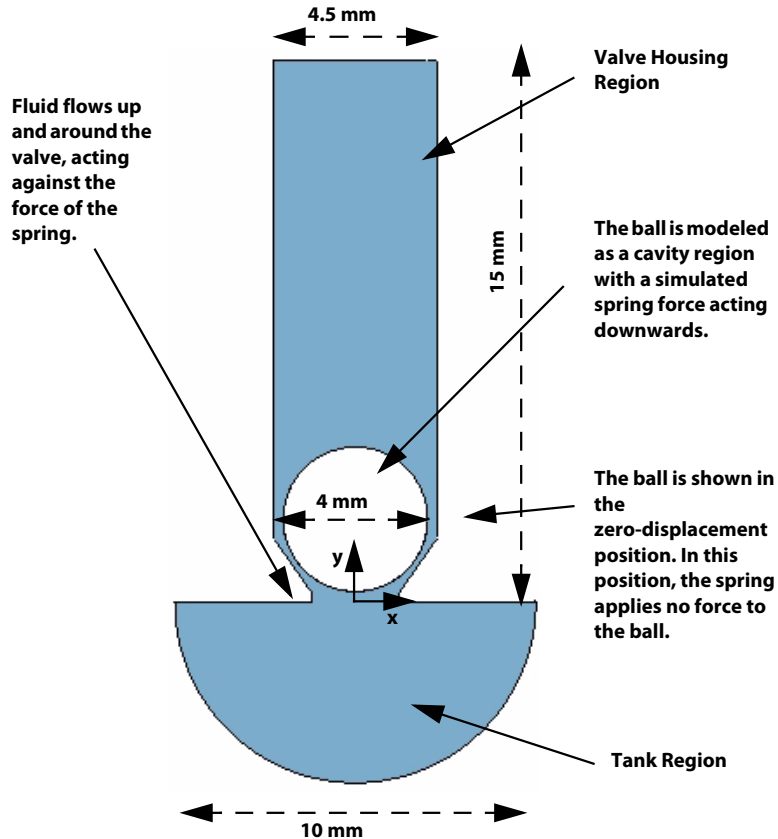
Component	Feature	Details	
ANSYS CFX-Pre	User Mode	General Mode	
	Simulation Type	Transient	
	Fluid Type	General Fluid	
	Domain Type	Single Domain	
	Turbulence Model	k-Epsilon	
	Heat Transfer	None	
	Output Control		
	CEL (CFX Expression Language)		
	User Fortran		
	Boundary Conditions		Opening
			Symmetry
			Wall
	Timestep	Transient	
Transient Results File			
ANSYS CFX-Post	Plots	Animation	
		Point	
		Slice Plane	
		Vector	
	Other	Opening	
		Symmetry Plane	
		Wall: No Slip	
	Wall: Moving		

In this tutorial you will learn about:

- Moving mesh
- Fluid-solid interaction (without modeling solid deformation)
- MPEG creation
- Monitoring points.

Overview of the Problem to Solve

This tutorial involves a moving mesh and a two-way fluid-structure interaction (FSI) between a ball and a fluid in a check valve. The geometry, modeled as a 2-D slice (0.1 mm thick), is displayed below.



Check valves are commonly used to allow uni-directional flow. The check-valve in this tutorial is located on the top of a tank, and acts as a pressure-relieving valve by moving to allow fluid to leave. The ball is connected to a spring that acts to push the ball downward when the ball is raised above the $y=0$ position. The forces on the ball are the force due to the spring and the force due to fluid flow. Gravity is neglected here for simplicity. The ball is represented as a cavity region in the mesh. The deformation of the ball is not modeled.

The tutorial is divided into two parts. In the first part, the motion of the ball is controlled by CEL expressions which account for the forces acting on the ball, including the force imparted by the flow. In the second part of the tutorial, the motion of the ball is controlled by a Junction Box Routine that updates the ball position at the start of each timestep by loading mesh coordinate files from a set of such files. The mesh coordinate files and required Fortran routines are provided with this tutorial.

Using CEL Expressions to Govern Mesh Deformation

The following topics will be discussed:

- [Setting up the Simulation in ANSYS CFX-Pre](#) (p. 334)
- [Modeling the Ball Dynamics](#) (p. 334)
- [Creating a New Simulation](#) (p. 335)
- [Importing the Mesh](#) (p. 335)
- [Importing the Required Expressions](#) (p. 336)
- [Setting the Simulation Type](#) (p. 336)
- [Creating the Domain](#) (p. 336)
- [Creating the Subdomain](#) (p. 337)
- [Creating the Boundary Conditions](#) (p. 338)
- [Setting Initial Values](#) (p. 339)
- [Setting Solver Control](#) (p. 340)
- [Setting Output Control](#) (p. 340)
- [Writing the Solver \(.def\) File](#) (p. 341)
- [Obtaining a Solution using the ANSYS CFX-Solver](#) (p. 341)
- [Viewing the Results in ANSYS CFX-Post](#) (p. 341)

Setting up the Simulation in ANSYS CFX-Pre

This section describes the step-by-step definition of the flow physics in ANSYS CFX-Pre.

Playing a Session File

If you wish to skip past these instructions, and have ANSYS CFX-Pre set up the simulation automatically, you can select **Session > Play Tutorial** from the menu in ANSYS CFX-Pre, then run the session file: `valveFSI.pre`. After you have played the session file as described in earlier tutorials under [Playing the Session File and Starting ANSYS CFX-Solver Manager](#) (p. 87), proceed to [Obtaining a Solution using the ANSYS CFX-Solver](#) (p. 341).

Modeling the Ball Dynamics

When defining your own simulations, the mesh motion may already be known. In such cases, it can be specified explicitly using the CEL. In this tutorial, the mesh motion is not known a-priori, and will be calculated using the forces that act on the ball. The dynamics equation that describes the motion of the ball is considered before setting up the simulation.

According to Newton's Second Law, the time rate of change in the ball's linear momentum is proportional to the net force acting on the ball. In differential form, the equation to be solved for the motion of the ball is:

$$m_{\text{Ball}} \frac{d}{dt}(\text{velBall}) = F_{\text{Flow}} - F_{\text{Spring}} \quad (\text{Eqn. 1})$$

where m_{Ball} is the mass of the ball (which is constant), vel_{Ball} is the velocity of the ball in the y coordinate direction, F_{Flow} is the flow (viscous and drag) force acting on the ball, and F_{Spring} is the spring force acting on the ball.

The left hand side of the equation is discretized to include an expression for the new displacement of the ball (relative to the spring's neutral position). The time derivative of the ball velocity is discretized as:

$$\frac{d(vel_{Ball})}{dt} = \frac{vel_{BallNew} - vel_{BallOld}}{tStep} \quad (\text{Eqn. 2})$$

where $vel_{BallNew}$ is further discretized as:

$$vel_{BallNew} = \frac{d_{BallNew} - d_{BallOld}}{tStep} \quad (\text{Eqn. 3})$$

The new displacement of the ball also appears in the expression for spring force:

$$F_{Spring} = k_{Spring} \times d_{BallNew} \quad (\text{Eqn. 4})$$

The discrete form of the equation of motion for the ball is re-assembled, and the ball displacement is isolated as:

$$d_{BallNew} = \frac{\left(F_{Flow} + \left(\frac{m_{Ball} \times vel_{BallOld}}{tStep} + \frac{m_{Ball} \times d_{BallOld}}{tStep^2} \right) \right)}{\left(k_{Spring} + \frac{m_{Ball}}{tStep^2} \right)} \quad (\text{Eqn. 5})$$

No further substitutions are required because all of these quantities are available through the CFX Expression Language as presented below.

Creating a New Simulation

1. Start ANSYS CFX-Pre.
2. Select **File > New Simulation**.
3. Select **General** and click **OK**.
4. Select **File > Save Simulation As**.
5. Under **File name**, type `ValveFSI`.
6. Click **Save**.

Importing the Mesh

1. Right-click `Mesh` and select **Import Mesh**.
2. Apply the following settings

Setting	Value
File type	PATRAN Neutral
File name	ValveFSI.out
Mesh Units	mm

3. Click **Open**.

Importing the Required Expressions


The expressions created in this step will determine the motion of the ball.

These expressions are provided in a CCL file. By importing the expression, you avoid the need to type each of the expressions by hand.

1. Select **File > Import CCL**.
This file is located in the working directory.
2. Select `ValveFSI_expressions.ccl`
3. Click **Open**.

You can review the imported expressions in the tree view.

Setting the Simulation Type

1. Click *Simulation Type* .
2. Apply the following settings

Tab	Setting	Value
Basic Settings	Simulation Type > Option	Transient
	Simulation Type > Time Duration > Total Time	tTotal
	Simulation Type > Time Steps > Timesteps	tStep
	Simulation Type > Initial Time > Time	0 [s]


3. Click **OK**.


Note: You may ignore the physics validation message regarding the lack of transient results files. You will set up transient results files later.

Creating the Domain

1. Right click *Simulation* in the **Outline** tree view and ensure that **Automatic Default Domain** is selected. 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
General Options	Basic Settings > Location	CV3D REGION, CV3D SUB
	Basic Settings > Domain Type	Fluid Domain
	Basic Settings > Fluids List	Methanol CH40*
	Domain Models > Pressure > Reference Pressure	1 [atm]
	Domain Models > Mesh Deformation > Option	Regions of Motion Specified
Fluid Models	Heat Transfer > Option	Isothermal
	Heat Transfer > Fluid Temperature	25 C

*. Click the **Ellipsis**  icon to open the **Fluids List** dialog box, then click *Import*

Library Data  to open the **Select Library Data to Import** dialog box. In that dialog box, expand *Constant Property Liquids* in the tree, then select *Methanol CH40* and click **OK**. Next, select *Methanol CH40* in the **Fluids List** dialog box and click **OK**.

- Expand **Mesh Motion Model** and ensure that **Mesh Stiffness > Option** is set to *Increase Near Small Volumes*.
- Click **OK**.

Mesh motion specifications are applied to two and three dimensional regions of the domain (that is, boundaries and subdomains, respectively) as follows:

- The mesh motion specification for the ball will be displacement in the y-direction according to the CEL expression *dBallNew* (which happens to be a single CEL variable).
- The mesh motion specification for the walls of the valve housing will be *Unspecified*. This settings 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. Since the ball moves vertically, the surrounding mesh nodes will also move vertically and will therefore remain on the valve housing. This mesh motion specification helps to preserve the quality of the mesh on the upper surface of the ball.
- The mesh motion specifications for the tank opening and tank volume will be *Stationary*.

The stationary tank volume ensures that the mesh does not fold at the sharp corner that exists where the valve joins the tank. The stationary mesh for the tank opening prevents the mesh nodes from moving (If the tank opening had unspecified mesh motion, the mesh nodes on this boundary would move vertically and separate from the non-vertical parts of the boundary.).

Creating the Subdomain

- Create a new subdomain named *Tank*.
- Apply the following settings

Tab	Setting	Value
Basic Settings	Location	CV3D SUB
Mesh Motion	Option	Stationary

3. Click **OK**.

Creating the Boundary Conditions

- Ball Boundary**
1. Create a new boundary condition named `Ball`.
 2. Apply the following settings

Tab	Setting	Value
Basic Settings	Boundary Type	Wall
	Location	BALL
Boundary Details	Wall Influence On Flow > Wall Velocity Relative To	(Selected)
	Wall Influence On Flow > Wall Velocity Relative To > Wall Vel. Rel. To	Mesh Motion
	Mesh Motion > Option	Specified Displacement
	Mesh Motion > X Component	0 [m]
	Mesh Motion > Y Component	dBallNew
	Mesh Motion > Z Component	0 [m]

3. Click **OK**.

Symmetry Boundary

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 condition named `Sym`.
2. Apply the following settings

Tab	Setting	Value
Basic Settings	Boundary Type	Symmetry
	Location	SYMP1, SYMP2

3. Click **OK**.

Vertical Valve Wall Boundary condition

1. Create a new boundary condition named `valveVertWalls`.
2. Apply the following settings

Tab	Setting	Value
Basic Settings	Boundary Type	Wall
	Location	VPIPE HIGHX, VPIPE LOWX

Tab	Setting	Value
Boundary Details	Wall Influence On Flow > Wall Velocity Relative To	(Selected)
	Wall Influence On Flow > Wall Velocity Relative To > Wall Vel. Rel. To	Boundary Frame
	Mesh Motion > Option	Unspecified

3. Click **OK**.

Tank Opening Boundary

1. Create a new boundary condition named `TankOpen`.
2. Apply the following settings

Tab	Setting	Value
Basic Settings	Boundary Type	Opening
	Location	BOTTOM
Boundary Details	Mass and Momentum > Option	Static Pres. (Entrain)
	Mass and Momentum > Relative Pressure	6 [atm]
	Turbulence > Option	Zero Gradient

3. Click **OK**.

Valve Opening Boundary

1. Create a new boundary condition named `ValveOpen`.
2. Apply the following settings


Tab	Setting	Value
Basic Settings	Boundary Type	Opening
	Location	TOP
Boundary Details	Mass and Momentum > Option	Static Pres. (Entrain)
	Mass and Momentum > Relative Pressure	0 [atm]
	Turbulence > Option	Zero Gradient
	Mesh Motion > Option	Stationary

3. Click **OK**.

Note: Opening boundary types are used to allow the flow to leave and re-enter the domain across the inflow and outflow boundaries. This behavior is expected due to the oscillatory motion of the ball and due to the potentially large region of flow re-circulation that will occur on the downstream side of the ball.

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]
	Initial Conditions > Cartesian Velocity Components > V	0.1 [m s ⁻¹]
	Initial Conditions > Cartesian Velocity Components > W	0 [m s ⁻¹]
	Initial Conditions > Static Pressure > Relative Pressure	0 [Pa]
	Initial Conditions > Turbulence Eddy Dissipation	(Selected)

3. Click **OK**.

Setting Solver Control

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


Tab	Setting	Value
Basic Settings	Transient Scheme > Option	Second Order Backward Euler
	Convergence Control > Minimum Number of Coefficient Loops	(Selected)
	Convergence Control > Minimum Number of Coefficient Loops > Min. Coeff. Loops	2*
	Convergence Control > Max. Coeff. Loops	5

*. This setting is optional. The default value of 1 is also acceptable.

3. Click **OK**.


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. Create a new transient result with the default name.
4. Apply the following settings to `Transient Results 1`

Setting	Value
Option	Selected Variables
Output Variables List	Pressure, Velocity
Output Frequency > Option	Time Interval
Output Frequency > Time Interval	tStep

5. Click the **Monitor** tab.
6. Select **Monitor Options**.
7. Under **Monitor Points and Expressions**:

- a. Click *Add new item* .
 - b. Set **Name** to `Ball Displacement`.
 - c. Set **Option** to `Expression`.
 - d. Set **Expression Value** to `dBallOld`.
8. Click **OK**.

Writing the Solver (.def) File

1. Click *Write Solver File* .
2. Apply the following settings:

Setting	Value
File name	ValveFSI.def
Quit CFX-Pre *	(Selected)

*. If using ANSYS CFX-Pre in Standalone Mode.

3. Ensure **Start Solver Manager** is selected and click **Save**.
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 folder if you have copied it there.
5. Quit ANSYS CFX-Pre, saving the simulation (`.cfx`) file at your discretion.

Obtaining a Solution using the ANSYS CFX-Solver

1. Ensure **Define Run** is displayed.
2. Click **Start Run**.
ANSYS 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. Click **Yes** to post-process the results.
4. If using Standalone Mode, quit ANSYS CFX-Solver Manager.

Viewing the Results in ANSYS CFX-Post

Creating User Locations and Plots

In the following steps, you will create an XY plane that lies midway between the two symmetry planes. The plane will be used to show mesh movement; it will also serve as a locator for a vector plot that will be used in an animation.


Creating a slice plane

1. Right-click a blank area in the viewer and select **Predefined Camera > View Towards -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	5e-05 [m]
Render	Draw Faces	(Cleared)
	Draw Lines	(Selected)





- Click **Apply** to create the plane.

Creating a Point




- Create a point at (0, 0.0003, 0.0001) using the XYZ method. This is a reference point for the low Y point of the ball at timestep 0. Click **Apply**.
- Click *Timestep Selector*  and load the results for a few different timesteps (for example: 0, 10, 20, 50, 90).
You will see the ball in different positions. The mesh deformation will also be visible. The maximum displacement occurs at around 20 timesteps (as was shown in the ANSYS CFX-Solver Manager), which is before the ball reaches equilibrium.

Creating an Animation with Velocity Vectors

You will create a vector plot and create an animation showing the velocity in the domain as the ball is displaced.

- Clear the visibility check box next to `Plane 1`.
- Create a vector plot, set **Locations** to `Plane 1` and leave **Variable** set to `Velocity`. Click **Apply**.
- Using the timestep selector, load time value 0.
- Click *Animation*  found in the toolbar.
The **Animation** dialog box appears.
- In the **Animation** dialog box:
 - Select **Keyframe Animation**.
 - Click *New*  to create `KeyframeNo1`.
 - Highlight `KeyframeNo1`, then change **# of Frames** to 148.
This will produce an animation keyframe at each timestep, resulting in an MPEG that plays for just over six seconds.
- Load the last timestep (150) using the timestep selector.
- 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.
 - Click *More Animation Options*  to expand the **Animation** dialog box.
 - Click the **Options** button.
 - On the **Options** tab, change **MPEG Size** to 720 x 480 (NTSC).
 - Click the **Advanced** tab, then set **Quality** to `Custom`.
 - Clear **Variable Bit Rate** and set **Bit Rate** to 3000000.

This limits the bit rate so that the movie will play in most players. You can lower this value if your player cannot handle this bit rate.

- g. Click **OK**.
- h. Select **Save MPEG**.
- i. Click *Browse*  next to the MPEG file name and set a file name for the MPEG file. If the file path is not given, the file will be saved in the working directory.
- j. Click **Save**. This sets the MPEG file name and path, but does not create the MPEG.
- k. Frame 1 is not loaded (The loaded frame is shown in the middle of the **Animation** dialog box, beside **F**). Click *To Beginning*  to load it, then wait a few seconds for the frame to load.
- l. Click *Play the animation* .

The MPEG is 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 MPEG file, you need to use a viewer that supports the MPEG format.

8. When you have finished, exit from ANSYS CFX-Post.

Using a Junction Box Routine to Govern Mesh Deformation

In this part of the tutorial, a Junction Box Routine will be used to read in a sequence of meshes, causing a sinusoidal motion of the ball. The meshes are provided for convenience; they were generated based on the following expression for displacement of the ball in the y direction as a function of time:

$$1 [\text{mm}] * (1 - \cos(2 * \pi * t / (20. * t\text{Step})))$$

This is an alternative to using CEL expressions to govern mesh deformation.

Important: You must have the required Fortran compiler installed and set in your system path in order to run this part of the 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.

The following topics will be discussed:

- [Setting up the Simulation in ANSYS CFX-Pre](#) (p. 344)
- [Creating a New Simulation](#) (p. 345)
- [Creating the Required Expressions](#) (p. 345)
- [Importing the Initial Mesh](#) (p. 346)
- [Defining the Junction Box Routine](#) (p. 346)
- [Setting Up the Junction Box Routine](#) (p. 347)
- [Setting the Simulation Type](#) (p. 347)
- [Creating the Domain](#) (p. 348)
- [Creating the Boundary Conditions](#) (p. 348)
- [Setting Initial Values](#) (p. 350)

- [Setting Solver Control](#) (p. 350)
- [Setting Output Control](#) (p. 350)
- [Writing the Solver \(.def\) File](#) (p. 351)
- [Obtaining a Solution using the ANSYS CFX-Solver](#) (p. 351)
- [Analyzing the Fluid Flow Force on the Ball](#) (p. 351)

Setting up the Simulation in ANSYS CFX-Pre

This section describes the step-by-step definition of the flow physics in ANSYS CFX-Pre.

Playing a Session File

If you wish to skip past these instructions, and have ANSYS CFX-Pre set up the simulation automatically, you can select **Session > Play Tutorial** from the menu in ANSYS CFX-Pre, then run the session file: `valveFSIUserF.pre`. After you have played the session file as described in earlier tutorials under [Playing the Session File and Starting ANSYS CFX-Solver Manager](#) (p. 87), proceed to [Obtaining a Solution using the ANSYS CFX-Solver](#) (p. 351).

Important: You must have the required Fortran compiler installed and set in your system path in order to run the session file successfully. 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.

Preparing the Working Directory

To prepare the working directory, copy the files and sub-directories contained in `<CFXROOT>/examples/valvefsi` into your working directory (Do not copy the `valvefsi` directory itself.).

The working directory should now contain the initial mesh file (`valveFSI.out`), plus two sub-directories. The `meshes` sub-directory contains meshes for one period of ball motion, with an amplitude of 1 mm, in the sequence of files `DefaultDomain.0` to `DefaultDomain.19`. The `junctionbox` sub-directory contains the Fortran source files that are used in the Junction Box Routine that will read the sequence of mesh files. The subroutines contained in these files are summarized as follows:

- `update_mesh_user`: Highest level Junction Box Routine that is responsible for replacing the mesh coordinates inside ANSYS CFX with the updated coordinates read in or defined by the low level routine, `set_mesh_user`.
- `set_mesh_user`: Low level routine that defines the updated mesh coordinates. In this tutorial, this is done by reading mesh files. In other applications, however, this could be done by using a set of Fortran commands that directly modify the existing mesh coordinates.
- `update_crdvx_user` and `upd_crdvx_user`: Routines to call for the generation of a node map between the initial mesh and the first user-defined mesh, and to repeatedly use this map to replace the mesh inside ANSYS CFX with the remaining sequence of user-defined meshes.

Two important attributes of the sequence of meshes read by the `SET_MESH_USER` routine warrant highlighting:

1. The coordinates of the first mesh in the sequence must be identical to the initial solver-internal mesh coordinates. This ensures that a node map between the user and initial solver-internal mesh coordinates can be generated.
2. The topology (i.e., connectivity) of all meshes in the sequence does not change. This ensures that the map between the user and solver-internal mesh coordinate can be re-used.


Creating a New Simulation

1. Start ANSYS CFX-Pre.
2. Select **File > New Simulation**.
3. Select **General** and click **OK**.
4. Select **File > Save Simulation As**.
5. Under **File name**, type `ValveFSIUserF`.
6. Click **Save**.


Creating the Required Expressions

The expressions created in this step are used for the simulation setup and for monitoring values during the solution process.


Force on the ball due to fluid flow The creation of this expression is very simple, as CFX Expression Language (CEL) provides a way to calculate directional force on any region.

1. Click *Expression* .
2. Set **Name** to `FFlow` then click **OK**.
3. Under **Definition**, enter the following expression:
`force_y()@Ball`
4. Click **Apply**.

Total simulation time The period of oscillation for the ball will be 1e-3 s (20 timesteps of 5e-5[s] each), and a total of two periods will be simulated.

1. Click *Expression* .
2. Set **Name** to `tTotal` then click **OK**.
3. Under **Definition**, enter the following expression:
`2e-3 [s]`.
4. Click **Apply**.

Timestep

1. Click *Expression* .
2. Set **Name** to `tStep` then click **OK**.
3. Under **Definition**, enter the following expression: `5e-5 [s]`
4. Click **Apply**.

Importing the Initial Mesh

1. On the **Outline** tab, right-click `Mesh` and select **Import Mesh**.
2. Change **File type** to `PATRAN Neutral`.
3. Select the file `valveFSI.out` and change **Mesh Units** to `[mm]`.
4. Click **Open**.

Defining the Junction Box Routine

In this part of the tutorial, you will run the `cfx5mkext` command to compile and link the provided Fortran routines. The resulting objects and libraries will be used later by ANSYS CFX-Solver.

Important: You must have the required Fortran compiler installed and set in your system path in order to run the `cfx5mkext` command successfully. 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.

1. Select **Tools > Command Editor**.
2. Type the following in the **Command Editor** dialog box (make sure you do not miss the semi-colon at the end of the line):

```
! system ("cfx5mkext -name meshread junctbox/*.F") < 1 or die;
```

- This is equivalent to executing the following at an OS command prompt:
`cfx5mkext -name meshread junctbox/*.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.
- `< 1 or die` will cause an error message to be returned if, for some reason, there is an error in processing the command.

3. Click **Process** to compile the subroutine.
4. Click **Clear** to remove all text from the command editor.

A subdirectory whose name is system dependent will be created in your working directory (For example, on IRIX a subdirectory named `irix` will be created in your working directory.). This subdirectory contains the shared object library named `meshread`.

Note: You can introduce the `-double` option to compile the subroutines for use with double precision ANSYS CFX-Solver executables.

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.


The following steps create a CCL object that specifies the path to the meshes directory and the number of meshes. The Fortran subroutine later looks up the values contained in this object so that it can determine where the meshes are located, and how many exist.

5. Type the following CCL into the **Command Editor** dialog box, replacing <filepath> with the path to your current directory.

```
USER:
MeshDir = <filepath>/meshes
NMeshes = 20
END
```

6. Click **Process**.


Setting Up the Junction Box Routine

1. Click *User Routine* .
2. Set **Name** to *Mesh Read* then click **OK**.
3. Apply the following settings

Tab	Setting	Value
Basic Settings	Option	Junction Box Routine
	Calling Name	update_mesh_user
	Library Name	meshread
	Library Path	(current working directory)
	Junction Box Location	Start of Time Step

4. Click **OK**.

Setting the Simulation Type

1. Click *Simulation Type* .
2. Apply the following settings

Tab	Setting	Value
Basic Settings	Simulation Type > Option	Transient
	Simulation Type > Time Duration > Option	Total Time
	Simulation Type > Time Duration > Total Time	tTotal
	Simulation Type > Time Steps > Option	Timesteps
	Simulation Type > Time Steps > Timesteps	tStep
	Simulation Type > Initial Time > Option	Automatic with Value
	Simulation Type > Initial Time > Time	0 [s]


3. Click **OK**.


Note: Instead of saving transient results files, you will use a monitor point to track the force on ball due to the flow. For this reason, you may ignore the physics validation message regarding the lack of transient results files.

Creating the Domain

1. Right click `Simulation` in the **Outline** tree view and ensure that **Automatic Default Domain** is selected. 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
General Options	Basic Settings > Location	CV3D REGION, CV3D SUB
	Basic Settings > Domain Type	Fluid Domain
	Basic Settings > Fluids List	Methanol CH40*
	Domain Models > Pressure > Reference Pressure	1 [atm]
	Domain Models > Mesh Deformation > Option	Junction Box Routine
	Domain Models > Mesh Deformation > Junction Box Routine	Mesh Read

*. Click the **Ellipsis**  icon to open the **Fluids List** dialog box, then click *Import*

Library Data  to open the **Select Library Data to Import** dialog box. In that dialog box, expand `Constant Property Liquids` in the tree, then select `Methanol CH40` and click **OK**. Next, select `Methanol CH40` in the **Fluids List** dialog box and click **OK**.

3. Click **OK**.

Creating the Boundary Conditions

Boundary for the ball

1. Create a new boundary condition named `Ball`.
2. Apply the following settings

Tab	Setting	Value
Basic Settings	Boundary Type	Wall
	Location	BALL
Boundary Details	Wall Influence on Flow > Option	No Slip
	Wall Velocity Relative To	(Selected)
	Wall Velocity Relative To > Wall Vel. Rel. To	Mesh Motion

3. Click **OK**.

Symmetry Boundary

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 condition named `Sym`.
2. Apply the following settings

Tab	Setting	Value
Basic Settings	Boundary Type	Symmetry
	Location	SYMP1, SYMP2

3. Click **OK**.

Tank Opening Boundary

1. Create a new boundary condition named `TankOpen`.
2. Apply the following settings

Tab	Setting	Value
Basic Settings	Boundary Type	Opening
	Location	BOTTOM
Boundary Details	Mass And Momentum > Option	Static Pres. (Entrain)
	Relative Pressure	6 [atm]
	Turbulence > Option	Zero Gradient

3. Click **OK**.

Valve Opening boundary

1. Create a new boundary condition named `valveOpen`.
2. Apply the following settings

Tab	Setting	Value
Basic Settings	Boundary Type	Opening
	Location	TOP
Boundary Details	Mass And Momentum > Option	Static Pres. (Entrain)
	Relative Pressure	0 [atm]
	Turbulence > Option	Zero Gradient

3. Click **OK**.

Note: Opening boundary types are used to allow the flow to leave and re-enter the domain across the inflow and outflow boundaries. This behavior is expected due to the oscillatory motion of the ball and due to the potentially large region of flow re-circulation that will occur on the downstream side of the ball.


Remaining Wall Boundary

1. On the **Outline** tab, double-click `Default Domain Default`.
2. Apply the following settings

Tab	Setting	Value
Boundary Details	Wall Influence on Flow > Option	No Slip
	Wall Velocity Relative To	(Selected)
	Wall Velocity Relative To > Wall Vel. Rel. To	Boundary Frame

3. Click **OK**.


Setting Initial Values

1. Click *Global Initialization* .
Since a transient simulation is being modeled, initial values are required for all variables.
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	0.1 [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 > Turbulence Eddy Dissipation	(Selected)
	Initial Conditions > Turbulence Eddy Dissipation > Option	Automatic with Value

3. Click **OK**.

Setting Solver Control

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

Tab	Setting	Value
Basic Settings	Transient Scheme > Option	Second Order Backward Euler
	Convergence Control > Minimum Number of Coefficient Loops	(Selected)
	Convergence Control > Minimum Number of Coefficient Loops > Min. Coeff. Loops	2*
	Convergence Control > Max. Coeff. Loops	5

*. This setting is optional. The default value of 1 is also acceptable.

3. Click **OK**.

Setting Output Control

A monitor point will be used to monitor the values of `FFLOW` at each timestep.

1. Click *Output Control* .
2. Click the **Monitor** tab.

3. Select **Monitor Options**.
4. Under **Monitor Points and Expressions**:
 - a. Click **New**.
 - b. Set **Name** to `force on ball due to flow`.
 - c. Set **Option** to `Expression` and **Expression Value** to `FFlow`.
5. Click **OK**.

Writing the Solver (.def) File

1. Click **Write Solver File** .

Note: Transient results files are not required for this specific simulation, so you may disregard the physics validation message and click **Yes** in the summary dialog box.

2. Apply the following settings:

Setting	Value
File name	ValveFSIUserF.def
Quit CFX-Pre*	(Selected)

*. If using ANSYS CFX-Pre in Standalone Mode.

3. Ensure **Start Solver Manager** is selected and click **Save**.
4. If using Standalone Mode, quit ANSYS CFX-Pre, saving the simulation (`.cfx`) file at your discretion.

Obtaining a Solution using the ANSYS CFX-Solver

When the ANSYS CFX-Solver Manager starts:

1. Click **Start Run**. The ANSYS CFX-Solver will calculate the solution.
2. Click **Yes** to post-process the results when the completion message appears at the end of the run.
3. If using Standalone Mode, quit ANSYS CFX-Solver Manager.

Analyzing the Fluid Flow Force on the Ball

In the ANSYS CFX-Solver Manager, click the **User Points** tab and observe the plot for the monitored value of `FFlow` (force imparted on the ball by the flow) as the solution develops. Using the mouse, click on various points on the curve.

Tutorial 21: Oscillating Plate with Two-Way Fluid-Structure Interaction

Introduction

This tutorial includes:

- [Tutorial 21 Features](#) (p. 354)
- [Overview of the Problem to Solve](#) (p. 354)
- [Setting up the Solid Physics in Simulation \(ANSYS Workbench\)](#) (p. 355)
- [Setting up the Fluid Physics and ANSYS Multi-field Settings in ANSYS CFX-Pre](#) (p. 358)
- [Obtaining a Solution using ANSYS CFX-Solver Manager](#) (p. 364)
- [Viewing Results in ANSYS CFX-Post](#) (p. 365)

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

- [Setting the Working Directory](#) (p. 1)
- [Changing the Display Colors](#) (p. 2)

Unless you plan on running a session file, you should copy the sample files used in this tutorial from the installation folder for your software (`<CFXROOT>/examples/`) to your working directory. This prevents you from overwriting source files provided with your installation. If you plan to use a session file, please refer to [Playing a Session File](#) (p. 358).

Sample files referenced by this tutorial include:

- `OscillatingPlate.pre`
- `OscillatingPlate.agdb`
- `OscillatingPlate.gtm`
- `OscillatingPlate.inp`

Tutorial 21 Features

This tutorial addresses the following features of ANSYS CFX.

Component	Feature	Details
ANSYS CFX-Pre	User Mode	General Mode
	Simulation 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 Details	Wall: Mesh Motion = ANSYS MultiField
Wall: No Slip		
Wall: Adiabatic		
Timestep	Transient	
ANSYS CFX-Post	Plots	Animation
		Contour
		Vector

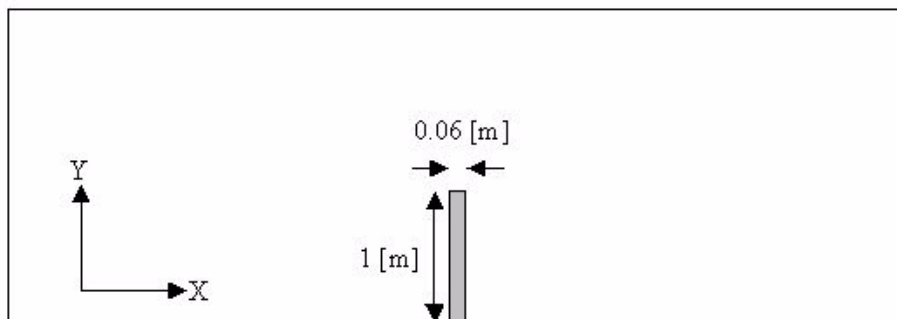
In this tutorial you will learn about:

- Moving mesh
- Fluid-solid interaction (including modeling solid deformation using ANSYS)
- Running an ANSYS Multi-field (MFX) simulation
- Post-processing two results files simultaneously.

Overview of the Problem to Solve

This tutorial uses a simple oscillating plate example to demonstrate how to set up and run a simulation involving two-way Fluid-Structure Interaction, where the fluid physics is solved in ANSYS CFX and the solid physics is solved in the FEA package ANSYS. Coupling between the two solvers is required throughout the solution to model the interaction between fluid and solid as time progresses, and 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. A thin plate 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 oscillations, which therefore have an amplitude that decreases in 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 solid deformation affects the fluid solution, and the fluid solution affects the solid deformation.

The tutorial describes the setup and execution of the calculation including the setup of the solid physics in Simulation (within ANSYS Workbench) and the setup of the fluid physics and ANSYS Multi-field settings in ANSYS CFX-Pre. If you do not have ANSYS Workbench, then you can use the provided ANSYS input file to avoid the need for Simulation.

Setting up the Solid Physics in Simulation (ANSYS Workbench)

This section describes the step-by-step definition of the solid physics in Simulation within ANSYS Workbench that will result in the creation of an ANSYS input file **OscillatingPlate.inp**. If you prefer, you can instead use the provided **OscillatingPlate.inp** file and continue from [Setting up the Fluid Physics and ANSYS Multi-field Settings in ANSYS CFX-Pre](#) (p. 358).

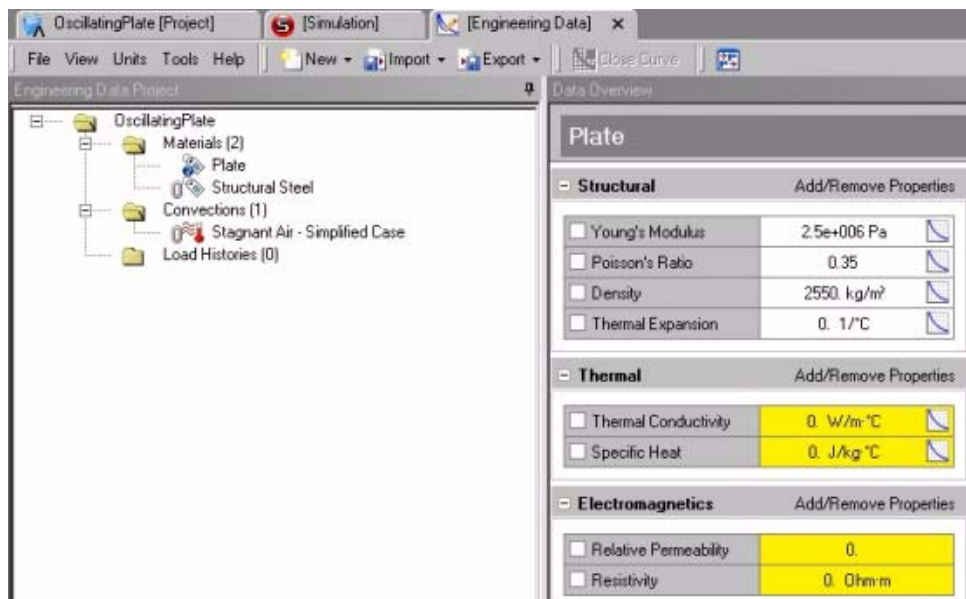
Creating a New Simulation

1. If required, launch ANSYS Workbench.
2. Click **Empty Project**. The **Project** page appears displaying an unsaved project.
3. Select **File > Save** or click **Save** .
4. If required, set the path location to a different folder. The default location is your working directory. However, if you have a specific folder that you want to use to store files created during this tutorial, change the path.
5. Under **File name**, type **OscillatingPlate**.

6. Click **Save**.
7. Under **Link to Geometry File** on the left hand task bar click **Browse**. Select the provided file `OscillatingPlate.agdb` and click **Open**.
8. Make sure that `OscillatingPlate.agdb` is highlighted and click **New simulation** from the left-hand taskbar.

Creating the Solid Material

1. When Simulation opens, expand `Geometry` in the project tree at the left hand side of the Simulation window.
2. Select `Solid`, and in the **Details** view below, select **Material**.
3. Use the arrow that appears next to the material name `Structural Steel` to select **New Material**.
4. When the **Engineering Data** window opens, right-click `New Material` from the tree view and rename it to `Plate`.



5. Enter 2.5×10^6 for **Young's Modulus**, 0.35 for **Poisson's Ratio** and 2550 for **Density**. Note that the other properties are not used for this simulation, and that the units for these values are implied by the global units in Simulation.
6. Click the **Simulation** tab near the top of the Workbench window to return to the simulation.

Basic Analysis Settings

The ANSYS Multi-field simulation is a transient mechanical analysis, with a timestep of 0.1 s and a time duration of 5 s.

1. Select **New Analysis > Flexible Dynamic** from the toolbar.
2. Select `Analysis Settings` from the tree view and in the **Details** view below, set **Auto Time Stepping** to `Off`.



3. Set **Time Step** to 0.1.
4. Under **Tabular Data** at the bottom right of the window, set **End Time** to 5.0 for the **Steps = 1** setting.

Inserting Loads

Loads are applied to an FEA analysis as the equivalent of boundary conditions in ANSYS CFX. In this section, you will set a fixed support, a fluid-solid interface, and a pressure load.

Fixed Support

The fixed support is required to hold the bottom of the thin plate in place.

1. Right-click **Flexible Dynamic** in the tree 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 selection.
3. Ensure **Fixed Support** is selected in the **Outline** view, then, in the **Details** view, select **Geometry** and click 1 **Face** to make the **Apply** button appear (if necessary). Click **Apply** to set the fixed support.

Fluid-Solid Interface

It is necessary to define the region in the solid that defines the interface between the fluid in CFX and the solid in ANSYS. Data is exchanged across this interface during the execution of the simulation.

1. Right-click **Flexible Dynamic** in the tree and select **Insert > Fluid Solid Interface** from the shortcut menu.
2. Using the same face-selection procedure described earlier, select the three faces of the geometry that form the interface between the solid and the fluid (low-x, high-y and high-x faces) by holding down <Ctrl> to select multiple faces. Note that this load is automatically given an interface number of 1.

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. Right-click **Flexible Dynamic** in the tree and select **Insert > Pressure** from the shortcut menu.
2. Select the low-x face for **Geometry**.
3. In the **Details** view, select **Magnitude**, and using the arrow that appears, select **Tabular (Time)**.
4. Under **Tabular Data**, set a pressure of 100 in the table row corresponding to a time of 0.

Note: 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 Simulation will automatically re-order the table in order of 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.

	Steps	Time	<input checked="" type="checkbox"/> 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.

Writing the ANSYS Input File

The Simulation settings are now complete. An ANSYS Multi-field run cannot be launched from within Simulation, so the **Solve** buttons cannot be used to obtain a solution.

1. Instead, highlight `Solution` in the tree, select **Tools > Write ANSYS Input File** and choose to write the solution setup to the file `OscillatingPlate.inp`.
2. The mesh is automatically generated as part of this process. If you want to examine it, select `Mesh` from the tree.
3. Save the Simulation database, use the tab near the top of the Workbench window to return to the **Oscillating Plate [Project]** tab, and save the project itself.

Setting up the Fluid 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 ANSYS CFX-Pre.

Playing a Session File

If you want to skip past these instructions and to have ANSYS CFX-Pre set up the simulation automatically, you can select **Session > Play Tutorial** from the menu in ANSYS CFX-Pre, then run the session file: `OscillatingPlate.pre`. After you have played the session file as described in earlier tutorials under [Playing the Session File and Starting ANSYS CFX-Solver Manager](#) (p. 87), proceed to [Obtaining a Solution using ANSYS CFX-Solver Manager](#) (p. 364).

Creating a New Simulation

1. Start ANSYS CFX-Pre.
2. Select **File > New Simulation**.
3. Select **General** and click **OK**.
4. Select **File > Save Simulation As**.
5. Under **File name**, type `OscillatingPlate`.

6. Click **Save**.

Importing the Mesh


1. Right-click `Mesh` and select **Import Mesh**.
2. Select the provided mesh file, `OscillatingPlate.gtm` and click **Open**.

Note: The file that was just created in Simulation, `OscillatingPlate.inp`, will be used as an input file for the ANSYS Solver.

Setting the Simulation Type

A transient ANSYS Multi-field run executes as a series of timesteps. The **Simulation Type** tab is used both to enable an ANSYS Multi-field run and to specify the time-related settings for it (in the **External Solver Coupling** settings). The ANSYS input file is read by ANSYS CFX-Pre so that it knows which Fluid Solid Interfaces are available.

Once the timesteps and time duration are specified for the ANSYS Multi-field run (coupling run), ANSYS 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`.

1. Click *Simulation Type* .
2. Apply the following settings

Tab	Setting	Value
Basic Settings	External Solver Coupling > Option	ANSYS MultiField
	External Solver Coupling > ANSYS Input File	OscillatingPlate.inp*
	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]
	Simulation Type > Option	Transient
	Simulation Type > Time Duration > Option	Coupling Time Duration
	Simulation Type > Time Steps > Option	Coupling Time Steps
	Simulation Type > Initial Time > Option	Coupling Initial Time


*. This file is located in your working directory.

3. Click **OK**.

Note: You may see a physics validation message related to the difference in the units used in ANSYS CFX-Pre and the units contained within the ANSYS input file. While it is important to review the units used in any simulation, you should be aware that, in this specific case, the message is not crucial as it is related to temperature units and there is no heat transfer in this case. Therefore, this specific tutorial will not be affected by the physics message.

Creating the Fluid

A custom fluid is created with user-specified properties.

1. Click *Material* .
2. Set the name of the new material to `Fluid`.
3. 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]
	Equation of State > Density	1 [kg m ⁻³]
	Transport Properties > Dynamic Viscosity	(Selected)
	Transport Properties > Dynamic Viscosity > Dynamic Viscosity	0.2 [Pa s]

4. Click **OK**.

Creating the Domain

In order to allow the ANSYS Solver to communicate mesh displacements to the CFX Solver, mesh motion must be activated in CFX.

1. Right click `Simulation` in the **Outline** tree view and ensure that **Automatic Default Domain** is selected. 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
General Options	Fluids List	Fluid
	Domain Models > Pressure > Reference Pressure	1 [atm]
	Domain Models > Mesh Deformation > Option	Regions of Motion Specified
Fluid Models	Heat Transfer > Option	None
	Turbulence > Option	None (Laminar)

3. Click **OK**.

Creating the Boundary Conditions

In addition to the symmetry conditions, another type of boundary condition corresponding with the interaction between the solid and the fluid is required in this tutorial.

Fluid Solid External Boundary

The interface between ANSYS and CFX is defined as an external boundary in CFX that has its mesh displacement being defined by the ANSYS Multi-field coupling process.

When an ANSYS Multi-field specification is being made in ANSYS CFX-Pre, it is necessary to provide the name and number of the matching **Fluid Solid Interface** that was created in Simulation. Since the interface number in Simulation 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 condition 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**.

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 condition 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 condition named `Sym2`.
5. Apply the following settings

Tab	Setting	Value
Basic Settings	Boundary Type	Symmetry
	Location	Sym2

6. Click **OK**.

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]
	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]

3. Click **OK**.

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, ANSYS and the data exchanged between the two solvers are all consistent. Within each stagger iteration, ANSYS and CFX 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 ANSYS, so ANSYS is set to solve before CFX 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 > Minimum Number of Coefficient Loops	(Selected)
	Convergence Control > Minimum Number of Coefficient Loops > Min. Coeff. Loops	2*
	Convergence Control > Max. Coeff. Loops	3
External Coupling	Coupling Step Control > Solution Sequence Control > Solve ANSYS Fields	Before CFX Fields

*. This setting is optional. The default value of 1 is also acceptable.

3. Click **OK**.


Setting Output Control

This step sets up transient results files to be written at set intervals.


1. Click *Output Control* .
2. On the **Trn Results** tab, create a new transient result with the default name.
3. Apply the following settings to *Transient Results 1*:

Setting	Value
Option	Selected Variables
Output Variable List	Pressure, Total Mesh Displacement, Velocity
Output Frequency > Option	Every Coupling Step*

*. This setting writes a transient results file every multi-field timestep.

4. Click the **Monitor** tab.
5. Select **Monitor Options**.
6. Under **Monitor Points and Expressions**:
 - a. Click *Add new item*  and accept the default name.
 - b. Set **Option** to *Cartesian Coordinates*.
 - c. Set **Output Variables List** to *Total Mesh Displacement X*.
 - d. Set **Cartesian Coordinates** to [0, 1, 0].
7. Click **OK**.

Writing the Solver (.def) File

1. Click *Write Solver File* .
2. If the **Physics Validation Summary** dialog box appears, click **Yes** to proceed.
3. Apply the following settings

Setting	Value
File name	OscillatingPlate.def
Quit CFX-Pre *	(Selected)

*. If using ANSYS CFX-Pre in Standalone Mode.

4. Ensure **Start Solver Manager** is selected and click **Save**.
5. If you are notified the file already exists, click **Overwrite**.
This file is provided in the tutorial directory and will exist in your working folder if you have copied it there.
6. Quit ANSYS CFX-Pre, saving the simulation (.cfx) file at your discretion.

Obtaining a Solution using ANSYS 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. ANSYS CFX-Solver Manager can be used to launch both solvers and to monitor the output from both.

1. Ensure the **Define Run** dialog box is displayed.
There is a new **MultiField** tab which contains settings specific for an ANSYS Multi-field simulation.
2. On the **MultiField** tab, check that the ANSYS input file location is correct (the location is recorded in the definition file but may need to be changed if you have moved files around).
3. 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 v110/ansys directory.
4. Click **Start Run**.

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. In this case, four lines appear, corresponding to two force components (FX and FY) and two displacement components (UX and UY). Since the analysis is 2D, FZ and UZ are not exchanged. 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.
2. When the solvers have finished and ANSYS CFX-Solver Manager puts up a dialog box to tell you this, click **Yes** to post-process the results.
3. If using Standalone Mode, quit ANSYS CFX-Solver Manager.

Viewing Results in ANSYS CFX-Post

For an ANSYS Multi-field run, both the CFX and ANSYS results files will be opened up in ANSYS CFX-Post by default if ANSYS CFX-Post is started from a finished run in ANSYS CFX-Solver Manager.

Plotting Results on the Solid

When ANSYS CFX-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 CFX-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 onto 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 **CFX-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 ANSYS CFX-Post, the Global Range is extended each time variable values are found outside the previous Global Range.

1. Turn on the visibility of `Boundary ANSYS` (under `ANSYS > Domain ANSYS`).
2. Right-click a blank area in the viewer and select **Predefined Camera > View Towards -Z**. Zoom into the plate to see it clearly.
3. Apply the following settings to `Boundary ANSYS`:

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, ANSYS CFX-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. Clear the visibility check box of `Boundary ANSYS`.
2. Create a contour plot, set **Locations** to `Boundary ANSYS` and `Sym2`, and set **Variable** to `Total Mesh Displacement`. Click **Apply**.
3. Using the timestep selector, load time value 1.1 [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 CFX-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**.



Creating an Animation


1. Using the **Timestep Selector** dialog box, ensure the time value of 0.1 [s] is loaded.

2. Clear the visibility check box 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. Select the visibility check box of `Boundary ANSYS`, 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:
 - a. Click *New*  to create `KeyframeNo1`.
 - b. Highlight `KeyframeNo1`, then change **# of Frames** to 48.
 - c. Load the last timestep (50) using the timestep selector.
 - d. Click *New*  to create `KeyframeNo2`.

The **# of Frames** parameter has no effect for the last keyframe, so leave it at the default value.
 - e. Select **Save MPEG**.
 - f. Click *Browse*  next to the MPEG file data box to set a path and file name for the MPEG file.

If the file path is not given, the file will be saved in the directory from which ANSYS CFX-Post was launched.
 - g. Click **Save**.

The MPEG file name (including path) will be set, but the MPEG will not be created yet.
 - h. Frame 1 is not loaded (The loaded frame is shown in the middle of the **Animation** dialog box, beside **F:**). Click *To Beginning*  to load it then wait a few seconds for the frame to load.
 - i. Click *Play the animation* .

The MPEG 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 MPEG file, you need to use a viewer that supports the MPEG format.
11. When you have finished, exit ANSYS CFX-Post.

Tutorial 22: Optimizing Flow in a Static Mixer

Introduction

ANSYS DesignXplorer (DX) 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 of the mixer and the input velocity of two fluids at two different temperatures into the mixer changes the effectiveness of the mixing. The measure of the mixing will be the output temperature.

DesignXplorer is a demonstration feature in this release. Refer to the ANSYS CFX Release Notes for more information about using DesignXplorer.

This tutorial includes:

- [Tutorial 22 Features](#) (p. 370)
- [Overview of the Problem to Solve](#) (p. 370)
- [Creating the Project](#) (p. 371)
- [Creating the Mesh](#) (p. 377)
- [Overview of ANSYS CFX-Pre](#) (p. 380)
- [Running Design Studies in DesignXplorer](#) (p. 387).

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

- [Setting the Working Directory](#) (p. 1)
- [Changing the Display Colors](#) (p. 2)

Sample files referenced by this tutorial include:

- **StaticMixerDX.pre**
- **StaticMixerDX.cmdb**

Tutorial 22 Features

This tutorial addresses the following features of ANSYS CFX:

Component	Feature	Details	
Design Modeler		Geometry Creation	
ANSYS CFX-Mesh		Mesh Creation Named Selections	
ANSYS CFX-Pre	User Mode	Quick Setup Wizard	
	Simulation 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
ANSYS DesignXplorer			
ANSYS CFX-Post			

In this tutorial you will learn about:

- Creating a geometry in DesignModeler and a mesh in CFX-Mesh.
- Using Quick Setup mode in ANSYS CFX-Pre to set up a problem.
- Using DesignXplorer to vary characteristics of the problem to see how you can improve the results.

Note: This tutorial assumes that you are familiar with the controls that enable you to change the orientation of the object in the Graphics window. If this is *not* the case, you should review [Tutorial 1: Simulating Flow in a Static Mixer Using CFX in Standalone Mode](#).

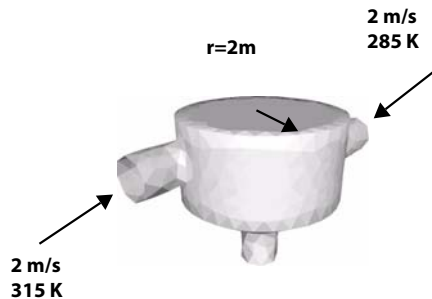
Overview of the Problem to Solve

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 entry is at a variable rate that has an initial value of 2 m/s and a temperature of 315 K. The second entry is at a variable rate that also has an initial value 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 ANSYS CFX DesignXplorer to optimize the amount of mixing of the water when it exits the static mixer, as measured by the water's temperature.

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



This tutorial begins by showing geometry creation and meshing. The following features are illustrated:

- Basic geometry creation, including Revolve and Extrude operations
- Basic meshing operations.

Creating the Project

To create the project:

1. In your file system, create a working directory for your project files.
2. Open ANSYS Workbench, go to **Start > All Programs > ANSYS 11.0 > ANSYS Workbench**.
Workbench displays the Start Page.
DesignXplorer is available only on Windows.
3. From the bottom of the **Start** dialog, under **Tools**, select **Options**.
4. Select **Common Settings > Geometry Import**.
Set the following four values to make named selections and parameters available to DesignXplorer:
 - **Parameter Processing** = Yes
 - **Personal Parameter Key** = <blank>
 - **Named Selection Processing** = Yes
 - **Named Selection Prefixes** = <blank>
5. Select **Meshing > CFX-Mesh Options > Volume Mesh** and confirm that **Volume Mesh Output** is set to ADD to Cmdb File.
6. Select **Meshing > Meshing** and set **Default Method** to CFX-Mesh.
7. Click **OK**.
8. Restart Workbench to apply the new settings.
9. When Workbench restarts, create a new project by selecting **Empty Project** from the **New** section of the **Start Page**. (Depending on which modules are available, you may find that you need to scroll down to see the **Empty Project** icon.)


The Project Page appears, containing an unsaved project.

10. The next step is to save your project. Select **File > Save As**, and save the project as `StaticMixerDX.wbdb` in your working directory. This directory will be the default location for all of the project file that you create during the tutorial.

Creating the Geometry


Now you can open DesignModeler in order to start creating the geometry.

Opening DesignModeler

1. In the left pane of the Project Page, click *New geometry*  (under **Create DesignModeler Geometry**).
DesignModeler loads and displays a window for selecting the desired length unit.
2. In the popup window, select **Meter** as the desired length unit and click **OK**. (Note that this window will not appear if you have previously set a default unit of measurement.)

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 draw a sketch of a cross-section and revolve it.

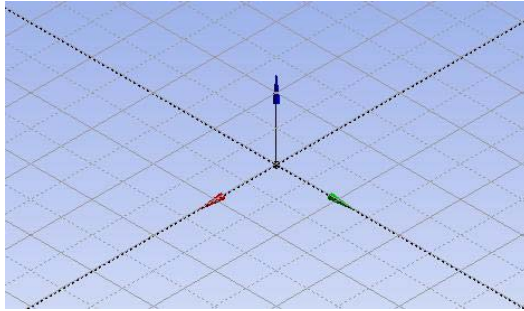
1. In the Tree Outline, click **ZXPlane**.
Each sketch is created in a plane. By selecting **ZXPlane** before creating a sketch, you ensure that the sketch you are about to create is based on the ZXPlane.
2. Click *New Sketch*  on the Active Plane/Sketch Toolbar, which is located above the Graphics window.
3. In the Tree Outline, click **Sketch1**.
4. Select the **Sketching** tab (below the Tree Outline) to view the available Sketching ToolBoxes.

Setting Up the Grid

Before starting your sketch, it helps to set up a grid of lines on the plane in which the sketch will be drawn. The presence of the grid allows 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**.

- To see the effect of changing **Minor-Steps per Major**, click, the right-mouse button to the top left of the plane center in the Graphics window and drag a box over the origin to zoom into the middle of the grid. When you release the mouse button, the model is magnified to show the selected area.



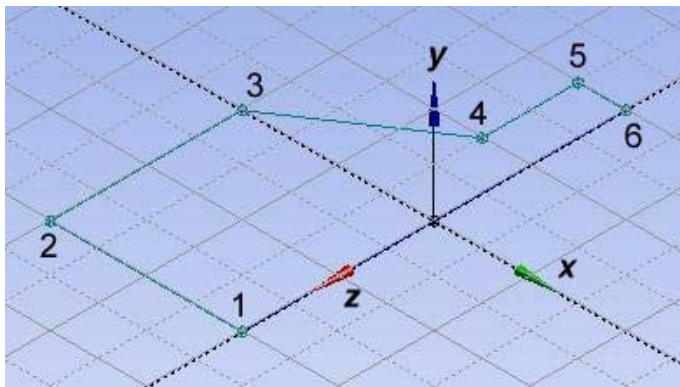
You now have a grid of squares with the smallest squares being 50 cm across. Because **Snap** is enabled, you can select only points that are on this grid to build your geometry; this can often help to position objects correctly.

Creating the Basic Geometry

Start by creating the main body of the mixer:



- From the **Sketching** tab, select the **Draw** Toolbox.
- Click **Polyline** and then create the shape shown below as follows:
 - Click on 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).
 - Click on each successive point to make the shape.
If at any point you click on the wrong place, click with the right-mouse button over the Graphics window and select **Back** from the shortcut menu to undo the point selection.
 - To close the polyline after selecting the last point, right-click and choose **Closed End** from the shortcut menu.

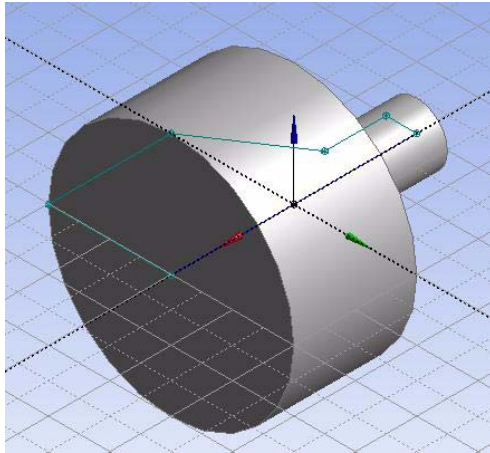
The information of the new sketch, **Sketch1**, appears in the Details View. Note that the longest straight line (4 m long) in the diagram below is along the z-axis (located at $x = 0$ m).



Revolving the Sketch


You will now create the main body of the mixer by revolving the new sketch around the Z-axis.

1. Click *Revolve*  from the tool bar above the Graphics window.
Details of the Revolve operation are shown in the Details View at the bottom left of the window. Leave the name of the Revolve as the default, **Revolve1**. The **Base Object** defines the sketch to be revolved. Select **Sketch1**.
2. In the **Details View** you should see **Apply** and **Cancel** buttons next to the **Axis** parameter; if those buttons are not displayed, click on the word **Axis**.
In the Graphics window, click on the grid line that is aligned with the X-axis of the plane represented by a red arrow (the X-axis of the plane is aligned with the global Z-axis), then click **Apply** in the Details View. The text next to **Axis** now changes to **Selected**.
3. Leave **Operation** set to **Add Material** because you need to create a solid. Ensure that **Angle** is set to **360** degrees and leave the other settings at their defaults.
4. To activate the revolve operation, click *Generate* . You can select this from the 3D Features Toolbar or from the shortcut menu by right-clicking in the graphics window.
After generation, you should find you have a solid as shown below.

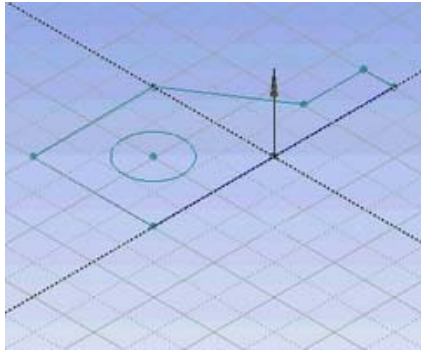


Create the First Inlet Pipe

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 on the plus sign next to **1 Part, 1 Body** to expand the tree structure.
2. Right-click on **Solid** and select **Hide Body**.
3. Select **ZXPlane** in the Tree Outline.
4. Click *New Sketch*  on the Active Plane/Sketch Toolbar.
5. Select the **Sketching** tab.
6. Create a circle as shown in the figure below:
 - a. Select the **Draw** Toolbox.
 - b. Select **Circle**, click on the grid to mark the center of the circle and then drag the mouse to define the radius. You may select an arbitrary radius.

7. Select the **Dimensions** Toolbox, select **General**, click on the circle in the sketch, then click near the circle to set a dimension. In the **Details View**, click the checkbox beside **D1**. When prompted, rename the parameter to `in1diameter` and click **OK**. This dimension will be a parameter that is modified in DesignXplorer.



Extrude the First Side-pipe

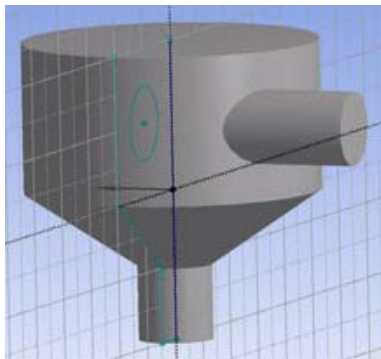
To create the first side-pipe extrude the sketch:

1. Click *Extrude*  from the 3D Features tool bar, located above the Graphics window.
2. In the Details View, change **Direction** to **Reversed**. Changing this parameter reverses the direction of the extrusion.
3. Change **Depth** to **3** (meters) and press Enter to set this value. All other settings can remain at their default values. The **Add Material** setting indicates that material is added to the existing solid, rather than a new solid being created.
4. To perform the extrude operation, click *Generate* .

Make the Solid Visible

To see the result of the previous operation, make the solid visible again:




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. A discrepancy with the size of the inlet pipe diameters may be visible but is not of concern.

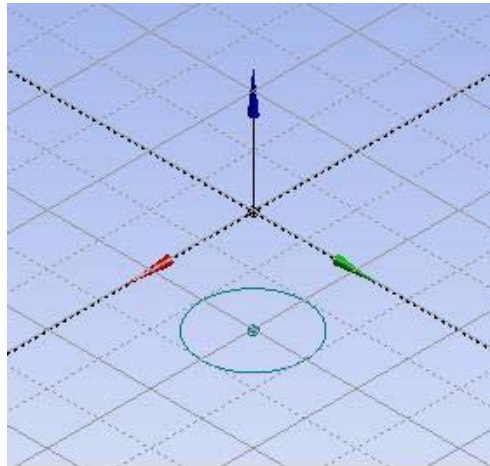



3. Right-click **Solid** and select **Hide Body**.


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. 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. Click the checkbox in the **FD1, Value 1** field. This sets the angle of rotation of this plane as a new design parameter. When prompted, set the name to `in2angle` and click **OK**.
5. Click **Generate** .
6. In the **Tree Outline** click on **Plane4**, then create a new sketch (sketch 3) based on this plane by clicking **New Sketch**  on the Active Plane/Sketch Toolbar.
7. Select the **Sketching** tab.
8. Click on **Settings** (in the **Sketching** tab) to open the **Settings** Toolbox.
9. Click on **Grid** and select **Show in 2D** and **Snap**.
10. Click on **Major Grid Spacing** and set it to **1**.
11. Click on **Minor-Steps per Major** and set it to **2**.
12. Right-click over the Graphics window and select **Isometric View** to put the sketch back into a sensible viewing position.
13. From the **Draw** Toolbox, select **Circle** and create a circle as shown below:





14. Select the **Dimensions** Toolbox, click **General**, click on the circle in the sketch, then click near the circle to set a dimension.
In the **Details View**, click the checkbox under **Dimensions**. When prompted to create a new design parameter, name the parameter `in2diameter` and click **OK**.
15. Click **Extrude**  from the 3D Features Toolbar.
16. In the Details View, ensure **Direction** is set to **Normal** in order to extrude in the same direction as the plane normal.
17. Ensure that **Depth** is set to **3** (meters). Leave the other settings at their defaults.

18. To activate the extrude operation, click *Generate* .
19. Right-click **Solid** in the Tree Outline and select **Show Body**.
The geometry is now complete.
20. Select **File > Save** and type in `StaticMixerDX.agdb`, then click **Save**.




Experiment with the Second Inlet Pipe

You can rotate the axis of the new plane manually to see the mixer with a range of different inlet angles for the second inlet:

1. In the **Tree Outline**, click on **Plane4**.
2. In the **Details View**, clear the check box in the **FD1, Value 1** field, then type in a value of 45 and click *Generate*  to see the new orientation of the second inlet.
3. Repeat the previous step with a value of -45.
4. Re-enable the check box in the **FD1, Value 1** field and click *Generate* ; the value automatically resets to 0.

Create Named Selections

To create the Named Selections:

1. Orient the static mixer so that you can see the inlet that has the lowest value of Y-coordinate. (You can rotate the mixer by holding down the middle-mouse button or the mouse scroll wheel.)
2. Highlight the inlet by left-clicking the inlet face, then right-click on the inlet and select **Named Selection**.
3. In the **Details View**, click **Apply**, then rename the **Named Selection** to: `in1`
4. Click *Generate* .
5. Orient the static mixer so that you can see the inlet that has the highest value of Y-coordinate.
6. Highlight the inlet by left-clicking the inlet face, then right-click on the inlet and select **Named Selection**.
7. In the **Details View**, click **Apply**, then rename the **Named Selection** to: `in2`
8. Click *Generate* .
9. Orient the static mixer so that you can see the outlet.
10. Highlight the outlet by left-clicking the outlet face, then right-click on the outlet and select **Named Selection**.
11. In the **Details View**, click **Apply**, then rename the **Named Selection** to: `out`
12. Click *Generate* .
13. Click **File > Save**.

Creating the Mesh

In order to create a mesh using CFX-Mesh, you take a geometry from DesignModeler then:

1. Define the Mesh Attributes.
2. Create the Surface Mesh (this is optional).

3. Create the Volume Mesh.

The creation of the surface mesh is optional; if it is not created explicitly, then it will be generated as part of the creation of the volume mesh.

Starting CFX-Mesh

To start CFX-Mesh:

1. At the top of the ANSYS Workbench window, click **StaticMixerDX [Project]** to return to the Project Page.
2. Ensure that the Design Modeler Geometry object (the **.agdb** file) is selected.
3. In the left pane, near the top under **DesignModeler Tasks**, click **New Mesh**. **[Meshing]** opens, then **[CFX-Mesh]** opens. At first glance, **[CFX-Mesh]** looks very similar to DesignModeler.

Define the Mesh Attributes

You will set a single size for all of the elements in this tutorial:

1. In the Tree View, in **Mesh > Spacing**, click **Default Body Spacing**.
2. In the Details View, confirm that **Maximum Spacing** is set to 0.3 m . This is a coarse length scale for this model, but is reasonable for a first run to generate an approximate solution and to test that the model is working correctly.

If you had to change the setting, select the new value, press Enter to set it, then press Enter again to commit the new value.

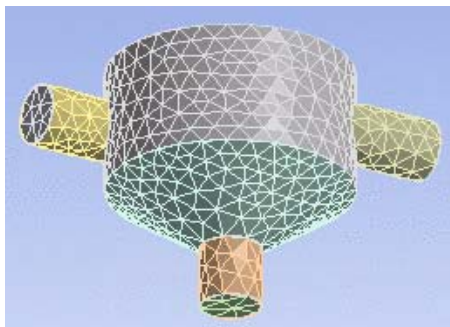
Generate the Surface Mesh

You will now have a look at the surface mesh to see the effect of the chosen length scale.

1. In the Tree View, click on the plus sign next to **Preview** to open it up.
2. Right-click **Default Preview Group** and select **Generate Surface Meshes**. The Default Preview Group always contains all faces in the geometry, so the mesh will be generated everywhere.

During the generation of the surface mesh, the progress displays in the status bar in the bottom right of the CFX-Mesh window.

Note: You can modify the way that the mesh is displayed by clicking on **Preview** in the Tree View and changing the options shown in the Details View. For example, by changing the Display Mode you can switch to display the mesh in Wire Mesh rather than with solid faces. Simply click on the name **Default Preview Group** to redisplay the surface mesh using the new settings.



It is not necessary to create the surface mesh within CFX-Mesh because it will be generated automatically when you create the volume mesh. However, it is generally a good idea to check the surface mesh before creating the volume mesh, to ensure that any settings you have made have the desired effect.

Generate the Volume Mesh

The volume mesh and all of the region information required for ANSYS CFX-Pre is stored in a Meshing file that has a `.cldb` file extension. This file is read into ANSYS CFX-Pre at the start of the simulation definition.

To generate the volume mesh:

1. In the Tree View, right-click on **Mesh** and select **Generate Volume Mesh**.

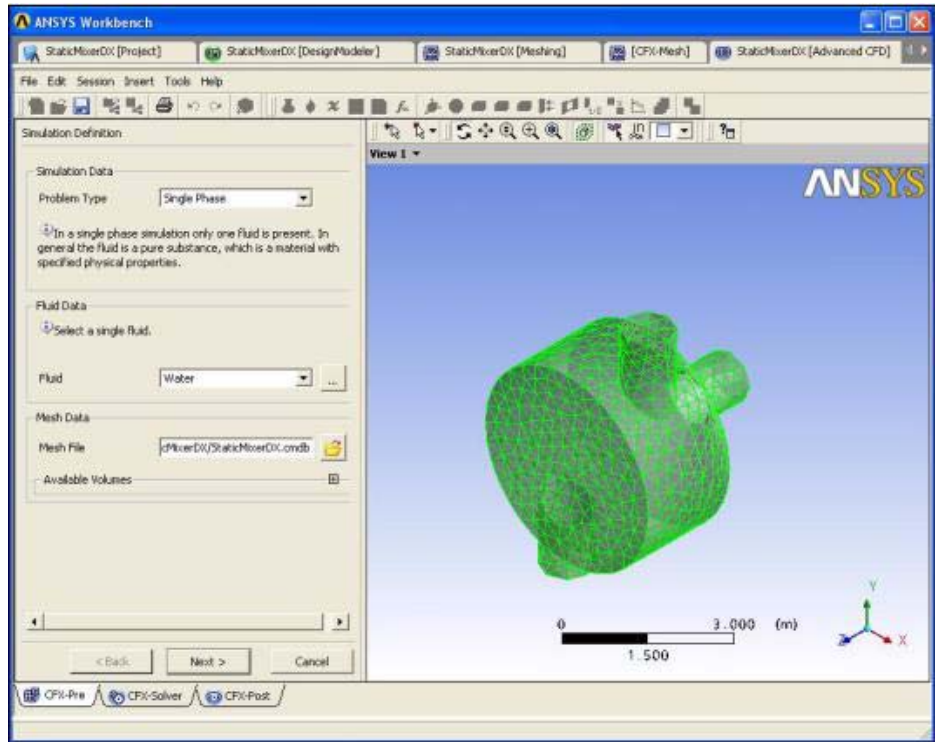
During the generation of the volume mesh, the progress displays in the status bar in the bottom right of the CFX-Mesh window. When the volume mesh is complete, the status bar disappears and you will be able to take control of the user interface again.

The mesh is now complete.

2. Select **File > Save** to save the meshing database as `StaticMixerDX.cldb` in the same directory as the other project files.
3. Click **StaticMixerDX [Project]** to return to the Project page.
4. Select **File > Save All** to save all the project files used in this tutorial.

Overview of ANSYS CFX-Pre

Because you are starting with the mesh you just created, you can immediately use ANSYS CFX-Pre to define the simulation. This is how ANSYS CFX-Pre looks with an imported mesh:



In the image above, the left pane of ANSYS CFX-Pre displays the **Simulation Definition**.

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).

Creating a New Simulation


Before importing and working with a mesh, a simulation needs to be started. You can define a simulation quickly by using Quick Setup mode:

1. Click on the StaticMixerDX workbench project (the **.wbdb** file on the project page), then in the left pane under **Advanced CFD**, click **Start CFX-Pre**.
After a few moments, the **[Advanced CFD]** tab appears.
2. Select **File > New Simulation**.
3. In the **New Simulation** dialog, select **Quick Setup** and click **OK**.
If an information popup appears, click **OK**.
4. Select **File > Save Simulation As**.
5. Under **File name**, type: `StaticMixerDX.cfx`
6. Click **Save**.

The tab name changes to **StaticMixerDX [Advanced CFD]**.

Setting the Physics Definition and Importing the Mesh

You need to specify the use of a prepared fluid, *Water*, which is defined to be water at 25°C, then select the mesh to import:

1. Ensure that **Simulation Definition** is displayed at the top of the Details View.
2. Under **Fluid Data > Fluid** select *Water*.
3. Under **Mesh Data > Mesh File**, click *Browse* .

The **Import Mesh** dialog box appears.
4. Under **File type**, select *ANSYS Meshing (*.cldb *.dsdb)*.
5. From your working directory, select *StaticMixerDX.cldb*.
6. Click **Open**.

The mesh loads.
7. Click **Next**.

Defining Model Data

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

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**.

Defining Boundaries

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

Define two inlet boundaries and one outlet boundary based on [Figure 1](#) (p. 371).

1. Ensure that **Boundary Definition** is displayed.
2. If *Inlet* is displayed, right-click and select **Delete**.
3. Right-click in the selector area, then select **New**.
4. Set **Name** to *in1*.
5. Click **OK**.



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

Setting Boundary Data

Once boundaries are created, you need to set the associated data. Based on [Figure 1](#) (p. 371), define the first inlet boundary:

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

Once boundary data is defined, the boundary needs to have the flow specification assigned. The first inlet boundary condition has a velocity that you will set to a variable called `in1Vel`.

1. Ensure that **Flow Specification** is displayed.
2. Set **Option** to `Normal Speed`.
3. Set **Normal Speed** to as follows:
 - a. Click on the input field to display the *Enter Expression* icon .
 - b. Click *Enter Expression* .
 - c. Type in: `in1Vel`

Once flow specification is defined, the boundary needs to have temperature assigned. Set the first inlet boundary condition to have a temperature of 315 [K] at one of the side inlets:

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

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	in1Vel
Temperature Specification	Static Temperature	315 [K]

Creating the Second Inlet Boundary Definition

Now that the first boundary has been created, the same concepts can be applied to specifying the second inlet boundary. Based on [Figure 1](#) (p. 371), you know the second inlet boundary condition consists of a velocity of `in2Vel` and a temperature of 285 K at the other of the side inlets. Define that now:

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

Tab	Setting	Value
Boundary Data	Boundary Type	Inlet
	Location	in2

Tab	Setting	Value
Flow Specification	Option	Normal Speed
	Normal Speed	in2Vel
Temperature Specification	Static Temperature	285 [K]

- There are no further boundary conditions that need to be set; click **Next**.

Moving to General Mode

You will now switch to general mode and make further adjustments to the simulation definition before saving the solver definition file.

- 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.
There are error messages about “Bad expression values”.
- When the **Outline** workspace appears, right-click `Default Domain` and select **Rename** from the shortcut menu.
- Set the name to `StaticMixerDX` and click **OK**.

Create the Expressions

You have parameters set to variables, but those variables have not yet been defined. This has resulted in error messages under the Graphics window.

Define the variables as expressions as follows:


- Create two new expressions named `in1Vel` and `in2Vel`:
 - In the **Tree Outline**, expand **Expressions, Functions and Variables** and right-click **Expression > Insert > Expression**.
 - Give the new expression the name: `in1Vel`
 - In the **Definition** area, type: `2 [m s^-1]`
Click **Apply**.
 - Right-click on **in1Vel** tab and select **Duplicate**. Name the new expression `in2Vel`.
When you click **OK**, notice that the error messages disappear.
- Right-click **in1Vel** and select **Use as DX Parameter**. Do the same for **in2Vel**. A small “P” appears on each expression’s icon.

Setting the Solver Controls

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 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**.

Writing the Solver (.def) File

You are now ready to write the definition file (which contains the information required by the ANSYS CFX-Solver for this CFD analysis), save the simulation file (**File** > **Save Simulation**) and start ANSYS CFX-Solver Manager.

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

- The simulation file can be saved at any time while defining the simulation.
- Mesh data is not contained in the simulation file; the simulation file references the original mesh file(s).

1. Click **File** > **Save Simulation** to save `StaticMixerDX.cfx`, the simulation file.

2. Click *Write Solver File* .

The **Write Solver File** dialog box is displayed.

3. If required, set **File name** to `StaticMixerDX.def`.
4. Ensure that `Start Solver Manager` is selected.
5. Click **Save**.
6. If you are notified the file already exists, click **Overwrite**.
7. If prompted, click **Yes** to save `StaticMixer.cfx`.

The definition file (`StaticMixer.def`) and the simulation file (`StaticMixer.cfx`) are created. ANSYS CFX-Solver Manager automatically starts and the definition file is set in the **Define Run** dialog box.

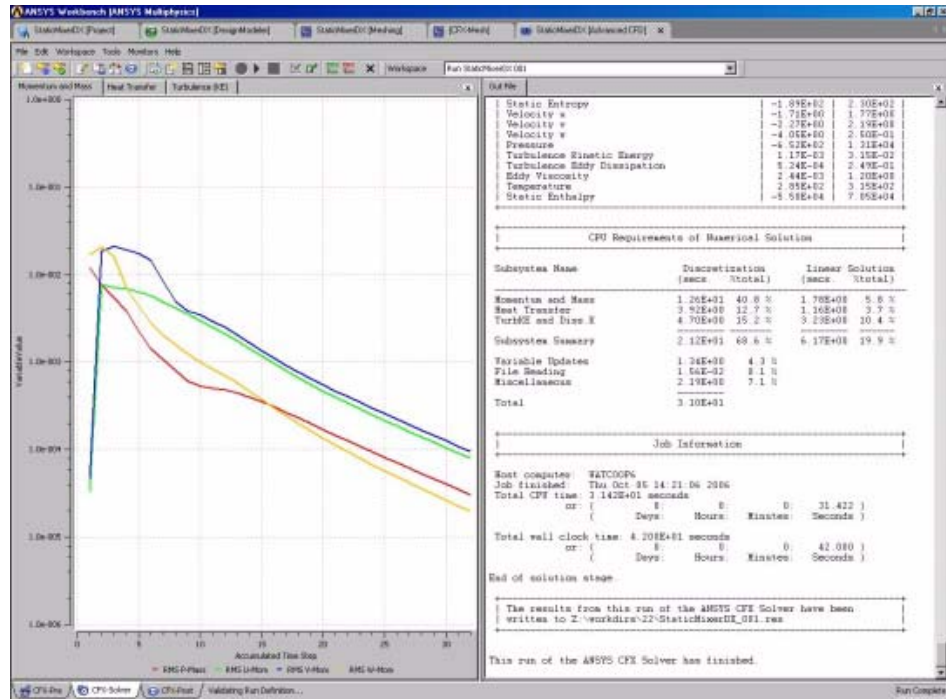
Starting the Run

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

When ANSYS CFX-Solver Manager is launched automatically from ANSYS 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 ANSYS CFX-Solver without having to define settings and values.

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

ANSYS 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 ANSYS 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.

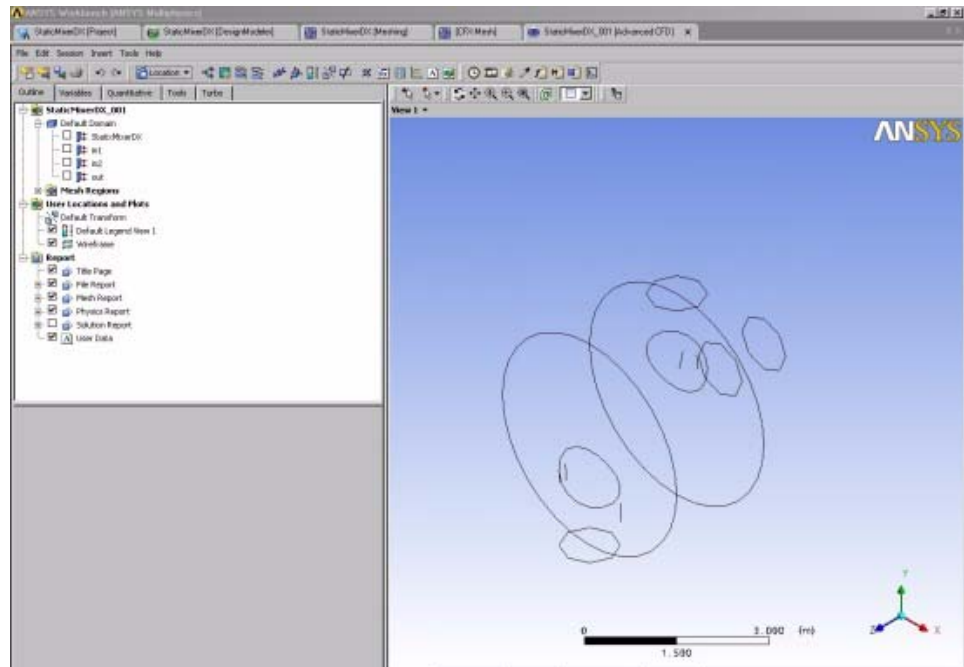
3. When ANSYS CFX-Solver is finished, click **Yes** to post-process the results.



Your project opens in CFX-Post.

Setting the Output Parameter in ANSYS CFX-Post

When ANSYS CFX-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 tool bar icons to manipulate the view, exactly as in ANSYS CFX-Pre.

You need to create an expression for the response parameter to be examined (Outlet Temperature) called OutTemp:

1. On the **Quantitative** tab, right-click on **Expressions > New**.
2. Type `OutTemp` and click **OK**.
3. In the **Definition** area:
 - a. Right-click **Functions > CFX-Post > areaAve**.
 - b. With the cursor between the parentheses, right-click and select **Variables > Temperature**.
 - c. Left-click after the @, then right-click and select **Locations > Outlet**.
 - d. Click **Apply**.
4. Open the **Expressions** branch. Right-click **OutTemp** and select **Use as DX Parameter**. A small "P" appears on the expression's icon.
5. Click **File > Save State As**. In the dialog that appears, ensure that `StaticMixerDX.cst` is the file name and click **Save**.

Important: The expression you created is written in CEL (CFX Expression Language). There is some CEL that works in CFX-Pre and CFX-Solver, but not in CFX-Post, so you should create all expressions for DesignXplorer output parameters in CFX-Post. Any expression created in CFX-Pre and used as a DesignXplorer output parameter could potentially cause fatal errors during the DesignXplorer run.

Running Design Studies in DesignXplorer

Create the DesignXplorer Info File

1. On the **StaticMixerDX[Project]** page, highlight the CFX-Post state file and click **Define CFX/DesignXplorer Study**.



The **Set up CFX/Design Explorer Study** dialog appears.

2. Select the **Specify Geometry Settings** checkbox. This enables you to vary both inlets sizes and the second inlet's orientation using the design parameters you created.
3. Specify the `.def` and state files that include the DX parameters selected earlier (these fields should already be filled).

For the **CFX/DX Info File**, input a name such as `TempStudy.cfdx` and click **Create CFX/DX Info File**.

4. A pop-up informs you that the file was written. Click **OK**.

DesignXplorer Studies

From the **DesignXplorer Startup Wizard** you can configure:

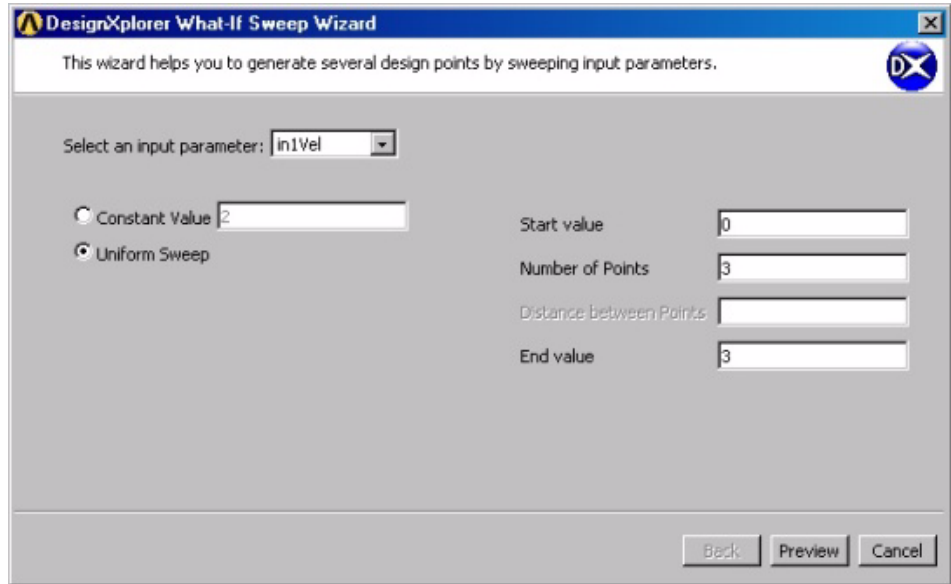
- A What-If Scenario
- A Deterministic Design of Experiments Scenario
- A Six Sigma Design of Experiments Scenario
- A Robust Design Scenario.

The sections that follow provide examples of the first two of these studies. For more information, refer to the DesignXplorer help that is available from the Workbench project's Help menu.

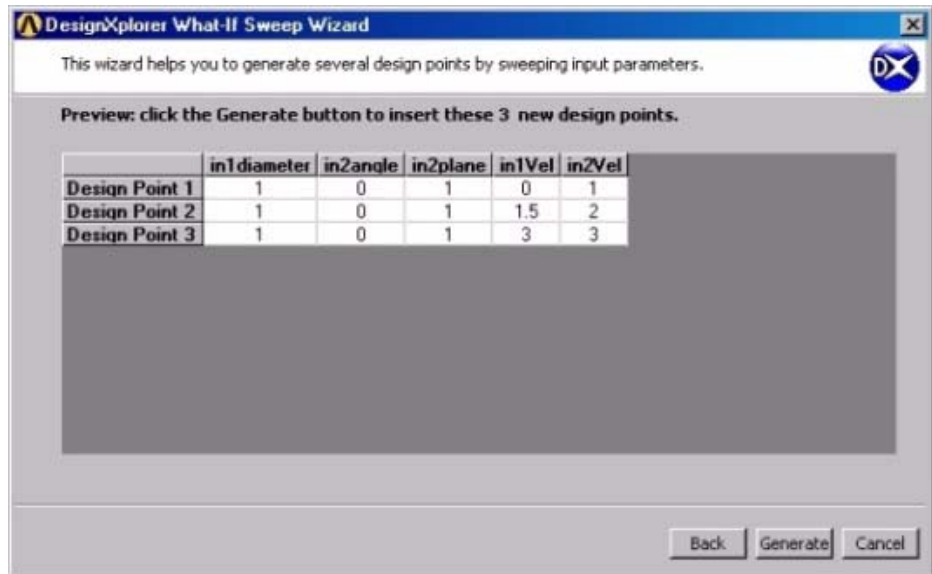
Configuring a What-If Scenario

1. On the **StaticMixerDX[Project]** page, highlight the `.cfdx` file and from the left pane click **Run CFX/DX Study**.
2. From the DesignXplorer Startup Wizard, choose **What-If**. Click **Finish**.
3. On the **What-If Design Points** page, click **Sweep Input Parameters**, which is near the bottom of the page.
4. Select the `in1Vel` input parameter from the dropdown list and select **Uniform Sweep**.
5. Specify the **Start Value** as **0** and the **End Value** as **3**.

- Specify the **Number of Points** = 3. (The number of CFD solver runs increases with number of points: with two input variables and three points each, nine solver runs are performed.)



- Select the in2Vel input parameter and repeat the previous steps, but choose a **Start Value** of 1. Setting one input velocity to zero as a design point gives you a chance to confirm that the simulation provides reasonable results: if in1Vel is 0, the output temperature should be roughly the same as the temperature of in2Vel (even at 0 input velocity, in1 does have a small affect on the water temperature in the mixer).
- Click **Preview**.



- Click **Generate** to insert the design points into the DX study.



10. Select **Run > Run Design Points**

The status bar at the bottom of the window shows the progress of the operation.

11. When the Solver runs are complete, post-process the results by clicking **Views > What-If Charts**.
12. In the left pane click **Views > Report** to ensure the outputs are reasonable.
13. Save the DesignXplorer study and click on the X to close DesignXplorer.

Configuring a Design of Experiments Scenario

1. On the **Project** page, highlight the `.cdfdx` file and from the left pane click **Run CFX/DX Study**.
2. From the DesignXplorer Startup Wizard, choose **Deterministic**, click **Next**, select **Design of Experiments**, and click **Finish**.
3. If asked to save your data, select **Yes**.
4. On the **Parameters** page, under **Parameter Properties**, enter a lower- and upper-bound for the input parameters. (Select the parameters in the left pane to do this.)
5. Click the lightning icon: **Run > Run Design Points**.



The status bar at the bottom of the window shows the progress of the operation.

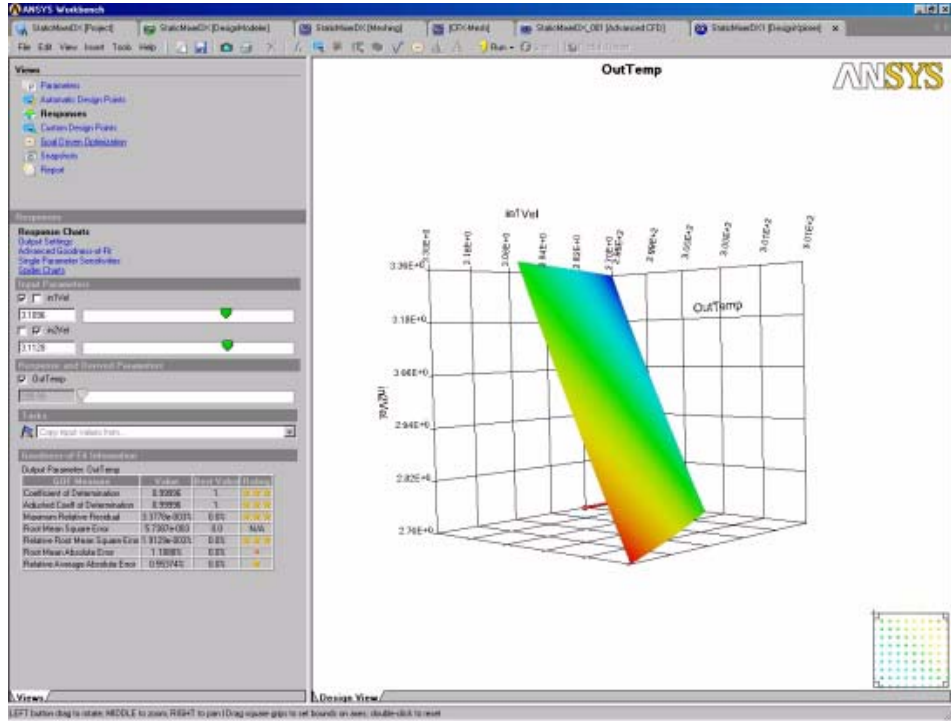
6. Once the solver runs have completed, click the **Save** icon to save the DesignXplorer Study.

Post-Process Results: Design of Experiments


Response Charts

1. Click **Views > Responses**, then **Responses > Response Charts**.

Here a response surface showing the variation of OutTemp with input various input parameters is displayed:

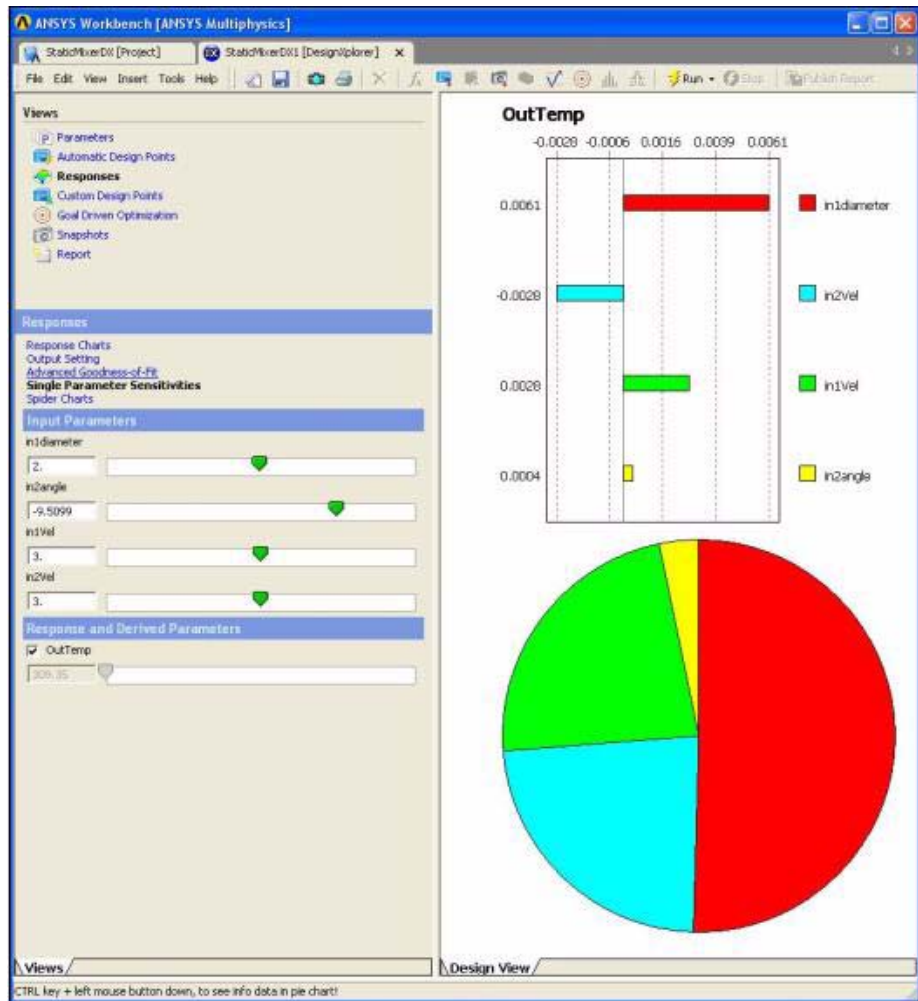


You can use the slide bars on the left to see how the output parameter varies with the changes you make to the input variables. This can be important when deciding the level of influence of input parameters over the response.


2. Create a figure (snapshot) for use in a report by clicking *Insert Snapshot*  .
3. Click **Views** > **Report** in the left pane to see the figure in a report.

Single-Parameter Sensitivity

From the left pane select **Views > Responses**, then **Responses > Single Parameter Sensitivities**. You can use the sliders to examine the sensitivity of the outlet temperature to each of the inlet conditions:



Min/Max Search

1. In the tool bar, click *Perform Min/Max Search* . The minimum and maximum values of the output parameter curvefit are returned.

Goal-Driven Optimisation

The objective in this set up is to maximise `OutTemp` while minimising the `in1diameter`.

1. Select **Goal Driven Optimisation** from the left pane.
2. In the right pane under **Input Parameter Goals**, set:
 - `in1diameter` **Desired Value** = Near Lower Bound, **Importance** = Higher

- OutTemp **Desired Value** = Maximum Possible, **Importance** = Higher.

Note: If you have chosen different parameter boundaries, the values in the graphic will differ from the values you see on your screen.

Input Parameter Goals

Click rows in this table to assign design goals to input parameters.

Name	Lower Bound	Upper Bound	Target	Desired Value	Importance
in1diameter	1.8	2.2		Near Lower Bound	Higher
in2angle	-11.	-9.		No Preference	Default
in1Vel	2.7	3.3		No Preference	Default
in2Vel	2.7	3.3		No Preference	Default

Response Parameter Goals

Click rows in this table to assign design goals to response parameters. Defining a target value is optional.


Name	Target	Desired Value	Importance	TradeOff
OutTemp	-	Minimum Possible	Default	On

Candidate Designs

 Generate or update candidate designs based on the current goals

3. In the **Sample Generation** section of the left pane, click **Generate**.
4. Click **Generate or update candidate designs based on current goals**.
A list of possible design solutions appears:

Candidate Designs

 Generate or update candidate designs based on the current goals

Parameter	Candidate A	Candidate B	Candidate C
in1diameter	1.814 ★★★	1.838 ★★	1.818 ★★★
in2angle	-9.49 --	-9.865 --	-10.74 --
in1Vel	2.7697 --	2.7252 --	2.9697 --
in2Vel	3.063 --	3.207 --	3.183 --
OutTemp	307.96 ★★	307.74 ★★	308.14 ★★

Another important analysis is to optimize certain values of your design to obtain a required response. The table above shows the recommended input values when trying to aim for a specific output when using certain input parameters.

5. Select the most appropriate design candidate and click **Insert selected candidate as a soft design**.
The chosen design is entered as a design point (under **Custom Design Points** in the left pane).
6. In the right pane under **Tasks**, click **Generate a hard reference design based on this design**.
The solver starts with boundary conditions supplied to match the current design point, and then a new reference design based on the soft design is inserted into the tree. You can then rate the chosen design and choose whether to accept or reject this candidate.
7. Repeat the process as necessary until a suitable final design can be selected.

Creating a Report

1. In the left pane, click **Report**. The **Design Parameters, Responses, Min/Max Search Results, Reference Design Points** (with their ratings), and any figures generated during post-processing all appear in the report.
2. Click **Publish Report** in the tool bar to output the report as an HTML file.

If You Make Changes to Parameters...

If after running DesignXplorer tests you make changes to the parameters in the `.cfx`, `.def`, or `.cst` files, you need to recreate the `.cfdx` file in order to update the Parameter Manager. See [Create the DesignXplorer Info File](#) (p. 387).

Exiting

When finished, select **File > Exit**. If prompted to save, select **No** (because you do not want to save the parameter data).

Tutorial 23: Aerodynamic & Structural Performance of a Centrifugal Compressor

Introduction

This tutorial includes:

- [Tutorial 23 Features](#) (p. 396)
- [Overview of the Problem to Solve](#) (p. 397)
- [Reviewing the Centrifugal Compressor Design](#) (p. 397)
- [Creating the Mesh in ANSYS TurboGrid](#) (p. 398)
- [Defining the Aerodynamic Simulation in ANSYS CFX-Pre](#) (p. 401)
- [Obtaining a Solution using ANSYS CFX-Solver Manager](#) (p. 403)
- [Viewing the Results in ANSYS CFX-Post](#) (p. 404)
- [Importing Geometry into DesignModeler](#) (p. 405)
- [Simulating Structural Stresses Due to Pressure Loads](#) (p. 406)
- [Simulating Structural Stresses Due to Rotation](#) (p. 407)

Note: This tutorial is intended to be run on Windows-based machines only.

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

- [Setting the Working Directory](#) (p. 1)

BladeModeler requires an ANSYS license to use, although BladeGen will run in "demonstration" mode without the license (no saving or exporting is possible in this mode). Please contact your ANSYS representative to obtain a license for BladeModeler, if you do not already have one. To learn how to configure your licenses, see *Configuring the BladeModeler License* in the *BladeModeler Help*.

Unless you plan on running a session file for ANSYS CFX-Pre, you should copy the sample files used in this tutorial from the installation folder for your software (<CFXROOT>/examples/) to your working directory. This prevents you from overwriting source files provided with your installation. If you plan to use a session file, please refer to [Playing a Session File](#) (p. 402).

If you want, you may skip the mesh creation sections and proceed directly to [Defining the Aerodynamic Simulation in ANSYS CFX-Pre](#) (p. 401), using the provided `Centrifugal_Compressor.gtm` file.

Sample files referenced by this tutorial include:

- `Centrifugal_Compressor.bgd`
- `Centrifugal_Compressor.gtm`
- `Centrifugal_Compressor.pre`

Tutorial 23 Features

This tutorial addresses the following features of ANSYS Workbench.

Component	Feature	Details
ANSYS BladeGen	Export	TurboGrid Input Files
ANSYS TurboGrid	Creating the Topology	
	Modifying the Topology	Modifying Control Points on Hub Modifying Control Points on Shroud
	Setting Mesh Parameters	Mesh Size Passage Inlet/Outlet
ANSYS CFX-Pre	User Mode	Turbomachinery
	Machine Type	Centrifugal Compressor
	Component Type	Rotating
	Simulation Type	Steady State
	Boundary Template	P-Total Inlet Mass Flow Outlet
	Flow Direction	Cylindrical Components
	Domain Type	Single Domain
	Timestep	Physical Timescale
ANSYS CFX-Post	Report	Computed Results Table
		Blade Loading Span 50
		Streamwise Plot of Pt and P
	Velocity Streamlines Stream Blade TE	
	Publish Report	
ANSYS DesignModeler	Create Blades	1
	Create Shroud Clearance	Relative Layer
	Create Fluid Zone	No

Component	Feature	Details
ANSYS Simulation	Static Structure	CFX Pressure
		Fixed Support
	Preview Surface Mesh	
	Static Structure Solutions	Von Mises Stresses
		Total Deformation

Overview of the Problem to Solve

The following tutorial demonstrates the scope of ANSYS software components which can be used for simulation of rotating machinery applications. A centrifugal compressor with 24 blades and shroud tip clearance, rotating about the z-axis is used as an example. This tutorial analyses both the aerodynamic and structural performance of the compressor.

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 BladeModeler. Once the mesh has been created, initial parameters defining the aerodynamic simulation will be set in ANSYS CFX-Pre and then solved in ANSYS CFX-Solver. The aerodynamic solution from the solver will then be processed and displayed in ANSYS CFX-Post.

To simulate structural stresses on the blade due to pressure loads from the aerodynamic analysis, as well as rotationally induced inertial effects, the existing geometry must be imported to ANSYS Simulation. Here these simulations can be defined and run. The final results will be animated to display the blade distortion.

Reviewing the Centrifugal Compressor Design

This section involves using BladeGen in ANSYS Workbench. BladeGen is a geometry creation tool specifically designed for turbomachinery blades. Since there is already a geometry created for this tutorial, this section will only require you to review the blade design.

1. Start ANSYS Workbench.
2. Click **Empty Project**.
The Project page appears.
3. Select **File > Save As** from the main menu and set an appropriate working directory for the simulation.
4. Save the project as **Centrifugal_Compressor.wbdb**.
5. Under **BladeGen**, click **Start BladeGen**.
6. When BladeGen opens select **File > Open** from the main menu and load the provided file, **Centrifugal_Compressor.bgd**.
7. Select **File > Export > TurboGrid Input Files** from the main menu. Ensure the working directory is set correctly and click **OK** to accept the default file names in the dialog box which appears.

Creating the Mesh in ANSYS TurboGrid

Using ANSYS TurboGrid a preliminary topology will be created and modified in order to eliminate overly large or small angles in the mesh. From this topology a final mesh will be created and imported into ANSYS CFX-Pre. If you wish, you can instead use the provided file, **Centrifugal_Compressor.gtm**, and continue from [Defining the Aerodynamic Simulation in ANSYS CFX-Pre](#) (p. 401)

Defining the Shroud Tip

For this compressor the shroud is stationary and a tip gap exists between the blade and shroud. To define this accordingly:

1. Return to the Project page and locate **Centrifugal_Compressor.inf**. Double-click the entry associated with this file to load the input file in ANSYS TurboGrid.
2. Expand **Geometry > Blade Set > Shroud Tip** and double click to edit.
3. Set **Clearance Type > Tip Option** to `Profile Number` and specify the tip profile number as 2. Click **Apply**.

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. Apply the following settings by opening `Topology Set` from the **Objects**:


Setting	Value
Topology Definition > Method	H/J/C/L-Grid
Include O-Grid > Width Factor	0.2
Tip Topology > Shroud	H-Grid Not Matching

2. Click **Apply**, then click **Freeze** to prevent unintended automatic changes to the topology.

Modifying the Topology

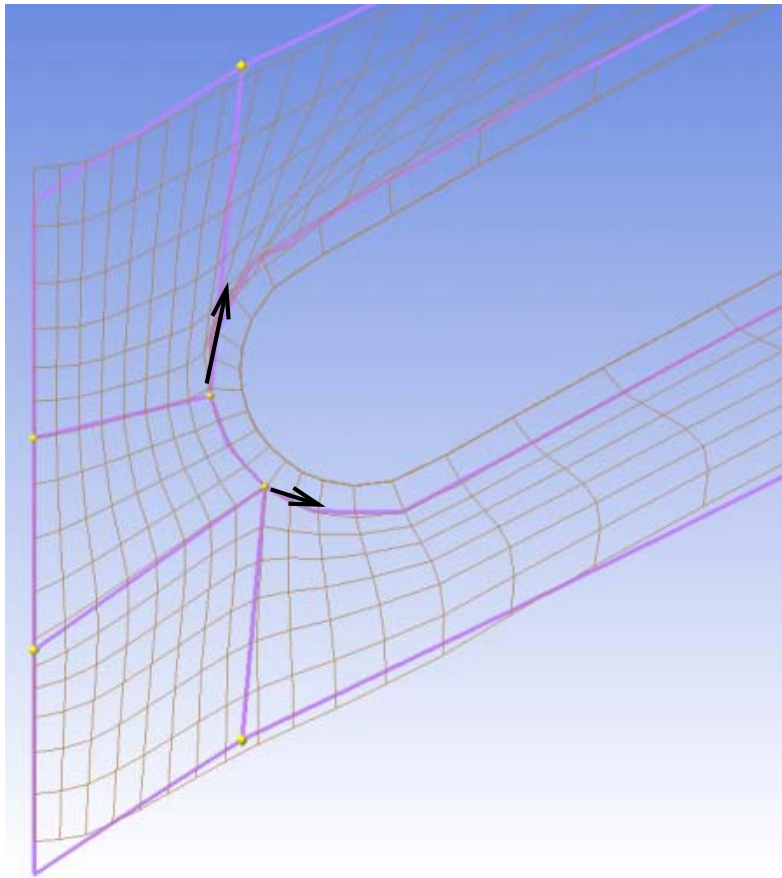
There are areas in the mesh which have small angles. These areas will be made more orthogonal by moving and inserting control points thereby increasing these angles.

Modifying Control Points on the Hub Layer

1. From the main menu, select **Display > Blade-to-Blade View > Use Passage Excluding Tip Transform**.
2. Right-click a blank area in the viewer and select **Transformation > Blade-to-Blade (Theta-M')**.
3. Click *Hide all geometry objects*  and clear the visibility check box next to `Layers > Shroud Tip` to make the hub topology more visible.
4. Open `Layers > Hub` in the details view and double-click `Minimum Face Angle` to view the problem areas of the mesh.

5. Zoom in on the area of the mesh near the leading edge shown in [Figure 1](#).
6. Select and move the control points as shown by the displacement vectors in [Figure 1](#).

Figure 1 **Modifying Control Points on the Hub**



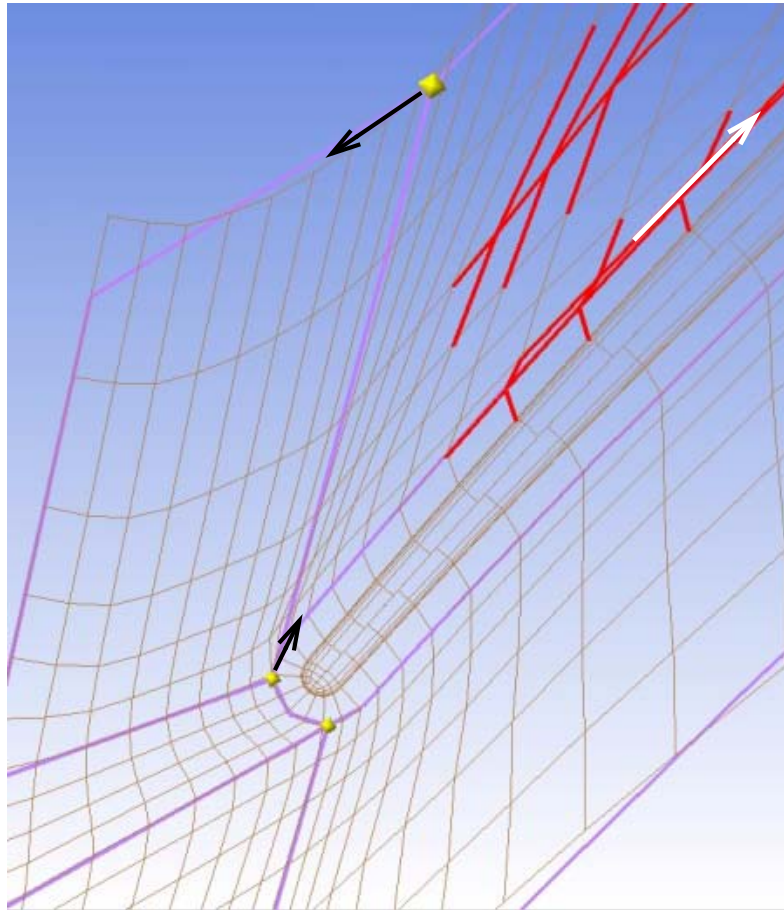
7. Confirm that the mesh statistics have improved for the `Hub` layer. Make further adjustments to the control points as necessary.

Modifying Control Points on the Shroud Layer

1. Clear the visibility check box of `Layers > Hub` and make `Layers > Shroud Tip` visible.
2. Open `Layers > Shroud Tip` in the details view and double-click `Minimum Face Angle` to view the problem areas of the mesh.

3. Select and move the two existing control points as shown by the black displacement vectors in Figure 2.

Figure 2 **Modifying Control Points on the Shroud**



4. Right click the area indicated by the tail of the white direction vector without a control point and select **Control Point > Insert Master**. Move this point in the direction and distance indicated.
5. Confirm that the mesh statistics have improved for the *Shroud* layer. Make further adjustments to the control points as necessary.

Setting up the Mesh Parameters

1. Open *Mesh Data* for editing under **Objects**.
2. Apply the following settings:

Tab	Setting	Value
Mesh Size	Node Count	Medium (100000)
	Inlet Domain	(Selected)
	Outlet Domain	(Selected)

Tab	Setting	Value
Passage	Spanwise Blade Distribution Parameters > Method	Element Count and Size
	Spanwise Blade Distribution Parameters > # of Elements	25
	Spanwise Blade Distribution Parameters > Const Element	11
	O-Grid > Method	Element Count and Size
	O-Grid > # of Elements	5
Inlet/Outlet	Inlet Domain > Override default # of Elements	(Selected)
	Inlet Domain > # of Elements	50
	Outlet Domain > Override default # of Elements	(Selected)
	Outlet Domain > # of Elements	25


3. Click **Apply**.

Note: Your mesh quality could decrease slightly after the node count is increased. If so, you might need to make minor adjustments to the hub and shroud control points to maintain the quality of your mesh before saving it and using it in the aerodynamic simulation that follows.

4. Open `Layers` for editing.
5. Right-click `Hub` and select **Insert Layer After** to insert a layer midway between the hub and shroud layers.
6. Select **File > Save State As** from the main menu and save the state as `Centrifugal_Compressor` in your working directory.

Creating and Saving the Mesh

With the topology and mesh data defined, the next step is to create and save a medium node count mesh based on this data.

1. Click *Create Mesh*  to create the mesh.
2. Select **File > Save Mesh As** from the main menu to open the **Save Mesh** dialog box.
3. Select **Combined in one domain, one file** under **Solver Type**.
This selection is required to properly format the mesh for subsequent steps in the Turbomachinery wizard in ANSYS CFX-Pre.
4. Ensure the working directory is set correctly and save the mesh file as `Centrifugal_Compressor.gtm`.

Defining the Aerodynamic Simulation in ANSYS CFX-Pre

After creating the mesh in ANSYS TurboGrid, it can be used in an aerodynamic simulation in ANSYS CFX. This section describes the step-by-step procedure for defining the simulation using the Turbomachinery wizard in ANSYS CFX-Pre.

Playing a Session File

If you wish to skip past these instructions, and have ANSYS CFX-Pre set up the simulation automatically, you can select **Session > Play Tutorial** from the menu in ANSYS CFX-Pre, then run the session file: `Centrifuga1_Compressor.pre`. After you have played the session file as described in earlier tutorials under [Playing the Session File and Starting ANSYS CFX-Solver Manager](#) (p. 87), proceed to [Obtaining a Solution using ANSYS CFX-Solver Manager](#) (p. 403).

Creating a New Simulation

This tutorial will use the Turbomachinery wizard in ANSYS CFX-Pre. This pre-processing mode is designed to simplify the setup of turbomachinery simulations.

1. Click on the Centrifugal_Compressor workbench project (the `.wbdb` file on the project page), then in the left pane under **Advanced CFD**, click **Start CFX-Pre**.
2. Select **File > New Simulation**.
3. Select **Turbomachinery** and click **OK**.
4. Select **File > Save Simulation As**.
5. Under **File name**, type `Centrifuga1_Compressor`.
6. Click **Save**.
7. In **Basic Settings** define **Machine Type** as **Centrifugal Compressor**.
8. Click **Next** to advance to **Component Definition**.

Component Definition

A rotor component needs to be defined using the mesh in the `Centrifuga1_Compressor.gtm` file.

1. In the blank area, right-click and select **New Component**.
2. Set the name to `ROTOR` and click **OK**.
3. Apply the following settings:

Setting	Value
Component Type > Type	Rotating
Component Type > Value	22360 [rev min ⁻¹]
Mesh > File	Centrifuga1_Compressor.gtm*
Mesh > Available Volumes > Volumes	Passage
Wall Configuration > Tip Clearance at Shroud	Yes

*. Select **CFX Mesh (*gtm *cfx)** as the File Type if `Centrifuga1_Compressor.gtm` is not visible.

4. Click **Next** to advance to the **Physics Definition** panel.

Physics Definition

In this section, you will set properties of the fluid domain and solver parameters.

1. Apply the following settings:

Setting	Value
Fluid	Air Ideal Gas
Simulation Type > Type	Steady State
Model Data > Reference Pressure	1 [atm]
Boundary Templates > P-Total Inlet Mass Flow Outlet	(Selected)
Boundary Templates > P-Total	0 [atm]
Boundary Templates > T-Total	20 [C]
Boundary Templates > Mass Flow	Per Component
Boundary Templates > Mass Flow Rate	0.167 [kg s ⁻¹]
Boundary Templates > Flow Direction	Cylindrical Components
Boundary Templates > Inflow Direction (a, r, t)	1, 0, 0
Solver Parameters > Convergence Control	Physical Timescale
Solver Parameters > Physical Timescale	0.0002 [s]

2. Click **Next** to advance to the **Interface Definition** panel.

Interface and Boundary Definition

The default settings for the next two panels are sufficient for this tutorial and need not be changed.

1. Click **Next** to advance to the **Boundary Definition** panel.
2. Click **Next** to advance to the **Final Operations** panel.

Final Operations

Now that the solver parameters and boundary conditions have been set, a definition file needs to be written to be used in ANSYS CFX-Solver Manager to solve for the associated physical variables.

1. Set **Operation** to `Start Solver` and enter `Centrifugal_Compressor` for **Solver File Name** then click **Finish**.

ANSYS CFX-Solver Manager opens.

Obtaining a Solution using ANSYS CFX-Solver Manager

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

When ANSYS CFX-Solver Manager is launched automatically from ANSYS 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 ANSYS CFX-Solver without having to define settings and values.

1. Click **Start Run**.

ANSYS 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 ANSYS CFX-Solver Manager operates.

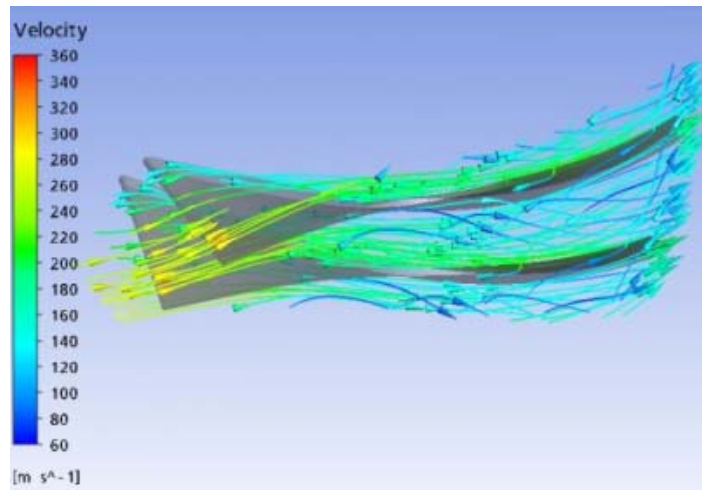
2. When ANSYS CFX-Solver is finished, click **Yes** in the dialog box that appears to post-process the results.
3. If using Standalone Mode, quit ANSYS CFX-Solver Manager.

Viewing the Results in ANSYS CFX-Post

ANSYS CFX-Post automatically post-processes the results from the solver that can be viewed in a number of ways including tables, and both 2-D and 3-D plots. The options for displaying individual results can be selected through the Outline tab while a full summary in the form of a report can also be generated and exported for viewing externally.

1. When ANSYS CFX-Post loads, a dialog box will appear prompting the load of the Centrifugal Compressor Rotor Report. Click **Yes**.
2. In the **Outline** tab under **Report** double click **Computed Results Table**.
This table presents measures of aerodynamic performance including required power and efficiencies.
3. In the **Outline** tab under **Report** 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.
4. In the **Outline** tab under **Report** double click **Streamwise Plot of Pt and P**.
This is a plot of the streamwise variation of pressure and total pressure.
5. In the **Outline** tab under **Report** double click **Velocity Streamlines Stream Blade TE View**.

This is a trailing edge view of the streamlines that start upstream of the blade.



6. To view a fully inclusive report, click on the **Report Viewer** tab found near the bottom right of the window. A report will be generated (this may take a few minutes) to include all figures available under **Report** in the **Outline** tab. This report can be viewed in ANSYS CFX-Post or exported to be viewed externally as an .html or .txt file.
7. To export this report, click on the **Publish** button in the toolbar just above the Report Viewer.
8. In the window that appears click **OK**. By default the format is .html and the location is your working directory under the name `CompressorRotorReport.html`. The report may take a few minutes to generate and save.
9. Select **File > Save State** to save the Post State file as `Centrifugal_Compressor.cst`.

Importing Geometry into DesignModeler

In order to perform static structural analysis of the compressor blade, the geometry must be imported into DesignModeler.

1. At the top of the ANSYS Workbench window, click **Centrifugal_Compressor [Project]** to return to the Project Page.
2. Ensure that the `Centrifugal_Compressor.wbdb` file is selected.
3. In the left pane, near the top under **Create DesignModeler Geometry**, click **New Geometry**.
4. Accept the default unit of **Millimeter**.
5. Click the **BladeEditor** button located in the top-right portion of the window.
6. Open the file `Centrifugal_Compressor.bgd`
7. In the **Tree Outline** select **ImportBGD1**.
8. In the Details view set **Create Hub** to **No**, **Create Blades** to **1**, **Create Shroud Clearance** to **Relative Layer** (to appropriately model the presence of the tip gap) and **Create Fluid Zone** to **NO** then click the **Generate** button in the toolbar.

Since we have specified that we did not want the hub to be created in the previous step, only the blade is shown in the viewer.

9. Save file as `Centrifugal_Compressor.agdb`

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. At the top of the ANSYS Workbench window, click **Centrifugal_Compressor [Project]** to return to the Project Page.
 2. Select **Tools > Options** from the main menu.
 3. Under **Meshing > Meshing > Meshing > Default Method** ensure that `CFX-Mesh` is selected and click **OK**.
 4. Make sure that `Centrifugal_Compressor.agdb` is selected then on the left panel click **New Simulation** under **DesignModeler Tasks**.
 5. Expand **Mesh** in the **Outline** view and click on **CFX-Mesh Method**.
 6. In the **Details** view below set **Definition > Method** to `Automatic`, then right-click on **Automatic Method** in the **Outline** view and select **Generate Mesh**.
 7. From the toolbar click on **New Analysis > Static Structural**.
 8. In the **Outline** right click on **Static Structural** and select **Insert > CFX Pressure**.
 9. Highlight the broad side of the blade in the viewer and click **Apply**, found next to **Geometry** in the **Details View**.
 10. Under **Definition** in the **Details View** choose **Import** for the **CFX Surface** option.
 11. If necessary browse to the CFX results file from your working directory and select **Rotor Blade** from the list of Surfaces and **0 s** from the list of Time Values. Click **OK**.
It may take a few seconds to apply these settings.
 12. Right click on **Static Structural** in the **Outline** and select **Insert > Fixed Support**.
 13. In the viewer, right-click on a blank area of the screen and select **View > Top**. Select the long thin face of the blade that is at the forefront of the displayed geometry and click **Apply**, next to **Geometry** in the Details section. This face is connected to the hub.
 14. Right click **Solution** under **Static Structural** in the **Outline** and select **Insert > Stress > Equivalent (von-Mises)**.
 15. From the toolbar click the **Solve** button.
When the solver has finished (this may take a few minutes) left click on **Static Structural > Solution > Equivalent Stress** to examine the results
 16. To animate the physical deformation of the blade along with the associated von-Mises Stress, click the **Play** button in the Animate window below the Geometry Viewer.
- Note:** You may need to drag the viewer window up in order to see the Animate window.
17. Right click on **Solution** under **Static Structural** in the **Outline** and select **Insert > Deformation > Total**.
 18. From the toolbar click the **Solve** button.

When the solver has finished (this may take a few minutes) left click on **Static Structural** > **Solution** > **Total Deformation** to examine the results.

19. The Animate window can be used to again animate the physical deformation.

Simulating Structural Stresses Due to Rotation

This section describes the steps required to simulate structural stresses on the blade due to pressure loads from the aerodynamic analysis as well as the inertial effects due to rotation.

1. Right click **Static Structural** from the **Outline** and select **Duplicate**.
2. Right click on the newly created object, `Static Structure 2` and rename it to `With-Rotation`
3. Making sure that **With-Rotation** is selected from the **Outline** click the **Inertial** button in the toolbar and select from its option's **Rotational Velocity**.
4. In the Details view for Rotational Velocity change **Define By** to **Components**.
5. Click on **Units** > **RPM** from the main menu.
6. In the Details view, enter a Z component of 22360 RPM.
7. Select **Solution** under **With-Rotation** from the **Outline** and click the **Solve** button on the toolbar.
8. To animate the Total Deformation or von-Mises Stress of the blade, select one found under **Static Structural** > **Solution** in the **Outline** and click the **Play** button in the Animate window below the Geometry Viewer.
9. Save simulation as `Centrifugal_Compressor.dsd` and return to the Project Page.
10. Click on **File** > **Save All** from the main menu and exit Workbench.

Index

Numerics

- 2D primitives
 - viewing 255

A

- additional variables
 - creating 228
 - setting 97
 - to model pH
 - creating 228
- airlift reactor example 269
- animation
 - plot animation 297
- ANSYS Field Solver (Structural) plot 364
- ANSYS Interface Loads (Structural) plot 365
- ANSYS Out File tab 365
- automotive catalytic converter tutorial 185
- axi-symmetric modelling example 223

B

- baffles example 253
- blade, impeller 253
- boundary condition profile file
 - creating 84
- boundary conditions
 - for free surface flows 142
 - modifying 101
- buoyancy
 - example 127
- butterfly valve example 165

C

- catalytic converter
 - automotive 185
 - example 185
- cavity example 127
- CEL limitations
 - create DX expressions in CFX-Post 386
- centrifugal compressor 395
- CFX-Mesh Options
 - configuring for DesignXplorer 371
- chemical reaction example 223
- CHT
 - example 239
- CHT example 239
- circular vent example 93
- combustion
 - calculating mass fractions 309, 316
 - eddy dissipation model 301
 - in a can combustor 299
 - laminar flamelet model 311
 - and multicomponent fluids 226
 - variable composition mixture 302
 - viewing concentrations 315
- combustion efficiency 309
- combustion models
 - loading multiple 316
- conjugate heat transfer
 - example 239
- Contour 318
- Contour plots 354
- contours
 - adding to surface plot 24, 52
- contours, adding 24, 52
- create
 - boundary conditions 113, 243, 338
 - fluid domain 189
 - isosurface 105

- pressure and volume fraction
 - expressions 142
- subdomain 190
- surface plot of y+ 123
- vectors 121
- creating and modifying streamlines 88

D

- default
 - legend 19, 47
- Define CFX/DesignXplorer study
 - how to 387
- Design of Experiments scenarios
 - configuring 387, 389
- design parameters
 - applying to a diameter 375
 - applying to a new plane 376
- DesignXplorer
 - configuring Workbench for 371
 - how to create Min/Max reports 392
 - static mixer optimization 369
- DesignXplorer parameter
 - creating a new 383
- DesignXplorer projects
 - use wbdb project files 372
- DesignXplorer studies
 - configuring 387, 389
- diameter
 - making into a design parameter 375
- domain
 - creating 231
- DX parameter
 - creating a new 383

E

- examples 1
 - 2D model 127
 - 2D modeling with 3D mesh 361
 - 3D region subdomain 337
 - airlift reactor 269
 - axi-symmetric 223
 - buoyancy 127
 - butterfly valve 165
 - calculating mass fractions 309, 316
 - catalytic converter 185
 - chemical reaction 223
 - CHT 239

- combustion eddy dissipation
 - model 301
- combustion efficiency 309
- combustion in a can combustor 299
- combustion models 311
- compiling a Fortran subroutine 287
- configuring moving mesh 337, 338
- conjugate heat transfer 239
- controlling the output of transient
 - results 218
- creating a boundary condition profile
 - file 84
- creating a profile boundary
 - condition 84
- creating a subdomain 190, 232, 243
- creating additional variables 228
- creating mesh adaption 146
- creating minimal transient results
 - files 103
- DesignXplorer 369
- discrete transfer radiation model 311
- exporting 2D stress 248
- Fluid-Structure Interaction (FSI) 333
- Fortran calling names, use lower-case
 - for 289
- Fortran subroutines, compiling 173
- free surface 139
- gas-liquid flow in an airlift reactor 269
- heat exchanger 239
- loading multiple combustion
 - models 316
- mixing tube 223
- Monte Carlo thermal radiation
 - model 289
- moving mesh, configuring 360, 361
- multicomponent flow 223
- multiphase flow 269
- P1 radiation model 301
- partitioned cavity 127
- radiation 310
- radiation in a can combustor 299
- radiation modeling 291
- radiation models 311
- radiation P1 model 301
- radiation properties 303
- setting a transient case in
 - animations 107
- setting additional variables 97
- setting options for a transient
 - scheme 350
- setting radiation flux 291

- setting radiation intensity 293
 - setting the transient scheme 340
 - setting up a transient simulation 347
 - solid region 239
 - static mixer 3, 31
 - static mixer optimization 369
 - steady state simulation 223, 239
 - supersonic flow 155
 - thermal radiation controls 314
 - thermal radiation modeling 291
 - thermal radiation models 312
 - transient animation, creating 221
 - transient ANSYS multi-field run 359
 - transient file in animation 107
 - transient mechanical analysis 356
 - transient results files, creating 293
 - transient results files, writing at intervals 363
 - transient results, configuring 133
 - transient rotor-stator 216
 - transient scheme 293
 - transient scheme, setting solver controls for 362
 - transient simulation 100, 127
 - transient simulation requires initial values 350
 - transient simulation type, configuring 130
 - transient simulation, enables mesh deformation 336
 - transient simulation, requires initial values 339, 362
 - transient simulation, uses Automatic With Value option 132
 - user CEL functions, using Fortran subroutines with 174
 - using CEL expressions with a moving mesh 334
 - valve 165
 - vent 93
 - viewing 2D primitives 255
 - viewing concentrations 315
 - wing 155
 - writing transient result files at intervals 340
 - expression
 - creating a new 386
 - expression language
 - shear rate dependent viscosity 202
 - velocity profile 172
 - expression method for inlet velocity profile 172
 - expressions
 - create in CFX-Post for DX 386
 - expressions to model the reaction creating 231
 - External Coupling
 - how to choose 362
- F**
- features
 - blade loading span 50 396
 - Centrifugal Compressor 396
 - CFX Pressure 397
 - computed results table 396
 - cylindrical components 396
 - fixed support 397
 - physical timescale 396
 - p-total inlet mass flow outlet 396
 - relative layer 396
 - steady state 396
 - streamwise plot of pt and p 396
 - total deformation 397
 - turbomachinery 396
 - velocity streamlines stream blade te 396
 - Von-Mises stress 397
 - Field Solver (Structural) plot 364
 - fixed support
 - defining 357
 - flow example
 - gas-liquid 269
 - multicomponent 223
 - multiphase 269
 - supersonic 155
 - fluid subdomain
 - creating 232
 - fluid-solid interactions 354
 - fluid-solid interface
 - defining 357
 - Fluid-Structure Interaction (FSI)
 - example 333
 - small mesh displacements in 366
 - Fortran calling names
 - use lower-case for 289
 - Fortran compiler
 - determining 173, 285, 287, 346
 - Fortran subroutine
 - compiling 287

- Fortran subroutines
 - compiling 173
- free surface
 - example 139
 - setting boundary conditions 142

G

- gas-liquid flow example 269
- generating output files 106
- Goal Driven Optimization
 - how to run 391

H

- heat exchanger example 239

I

- impeller blade example 253
- inlet (supersonic) 158
- Interface Loads (Structural) plot 365

M

- master control point 400
- mesh adaption
 - creating 146
- mesh deformation
 - requires transient simulations 336
- mesh displacements
 - magnifying 366
- method for the inlet velocity profile, expression 172
- mixer
 - static mixer example 3, 31, 369
- mixing tube example 223
- model
 - creating 61
- modelling example
 - 2D 127
 - axi-symmetric 223
- modify
 - streamlines 88
- moving mesh
 - configuring 337, 338, 360, 361
 - example 333
 - using CEL expressions with 334

- multicomponent flow example 223
- multiphase
 - flow example 269
- multiphase mixer example 251

N

- Named Selection Prefixes
 - configuring for DesignXplorer 371
- Named Selection Processing
 - configuring for DesignXplorer 371
- new plane
 - creating as a design parameter 376

O

- obtaining a solution
 - in parallel 117
 - in serial 116
- outlet (supersonic) 159
- outline plot 16, 44
- output files
 - generating 106

P

- P1 radiation model 301
- parallel
 - running 116
- parallel solution example 219
- Parameter Processing
 - configuring for DesignXplorer 371
- Personal Parameter Key
 - configuring for DesignXplorer 371
- pH, calculating 230
- power syntax 125
- pressure
 - defining 357
- printing greyscale 309
- profile boundary condition
 - creating 84
- project
 - creating in Workbench (DesignXplorer) 371
- publish report 405

R

- radiation
 - in a can combustor 299
 - modeling at a window 291
 - setting a Monte Carlo thermal model 289
 - viewing 310
- radiation flux
 - setting 291
- radiation intensity
 - setting 293
- radiation models
 - discrete transfer 311
- radiation properties
 - setting 303
- reaction
 - defining 229
- run
 - in parallel 116
 - monitoring 220
- Run Design Points
 - how to 389

S

- set
 - boundary conditions 290, 304
 - buoyancy reference density 143
 - initial values 161
 - transient rotor-stator calculation 216
- simulation example
 - steady state 223, 239
 - transient 127
- simulations
 - creating a solid material in 356
 - creating solid physics in 355
- solid
 - region example 239
- solid deformations
 - modeling 354
- solvers
 - coupling two to model interactions 354
- stagger iterations 362
- static
 - mixer example 3, 31
- steady state simulation example 223, 239
- streamlines
 - creating and modifying 88
- structural properties
 - defining 356

- subdomain
 - a 3D region 337
 - creating 190, 232, 243
- superficial velocity 281
- supersonic flow
 - example 155
- surface plot 124
- syntax, power 125

T

- text
 - auto-annotation 106, 297
- thermal radiation
 - modeling at a window 291
- thermal radiation controls
 - setting 314
- thermal radiation models
 - setting 312
- transient animation
 - creating 221
- transient ANSYS multi-field run
 - executes as time steps 359
- transient case
 - setting in animations 107
- transient file
 - used in animations 107
- transient mechanical analysis
 - example 356
- transient result files
 - writing at intervals 340
- transient results
 - configuring 133
- transient results files
 - creating 293
 - creating minimal 103
 - writing at intervals 363
- transient rotor-stator calculation 216
- transient scheme
 - setting 293, 340
 - setting solver controls for 362
- transient scheme, setting options for 350
- transient simulation
 - enables mesh deformation 336
 - initial values required 350
 - modifying the domain for 217
 - requires initial values 362
 - setting up 347
- transient simulation type
 - configuring 130

- defining 100
- transient simulation, uses Automatic With Value option 132
- transient simulations
 - example 93, 127
 - require initial values 339
- tutorial examples 1
 - see also examples*
- two-dimensional modelling example 127

U

- user CEL functions
 - using Fortran subroutines with 174
- user function names
 - must differ from user routine names 175
- user routine names
 - must differ from user function names 175
- using
 - cfx5mkext command 173, 288, 346
 - symmetry planes 119

V

- valve
 - example 165
- variables
 - user vector 163
- velocity
 - superficial 281
- vent example 93
- viewing
 - inflated elements 72
 - mesh partitions (parallel only) 125
 - results 149

W

- wall
 - boundary conditions 292, 306
 - free slip 113
- wbdb files
 - using with DesignXplorer projects 372
- What-If scenarios
 - configuring 387, 389
- wing example 155
- Workbench
 - configuring for DesignXplorer 371
 - starting on Windows 371