

# **ANSYS Fluent Tutorial Guide**



ANSYS, Inc. Southpointe 2600 ANSYS Drive Canonsburg, PA 15317 ansysinfo@ansys.com http://www.ansys.com (T) 724-746-3304 (F) 724-514-9494

Release 18.0 January 2017



#### **Copyright and Trademark Information**

© 2016 SAS IP, Inc. Unauthorized use, distribution or duplication is prohibited.

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

#### **Disclaimer Notice**

THIS ANSYS SOFTWARE PRODUCT AND PROGRAM DOCUMENTATION INCLUDE TRADE SECRETS AND ARE CONFID-ENTIAL 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: 2008 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 legal information in the product help files for the complete Legal Notice for ANSYS proprietary software and third-party software. If you are unable to access the Legal Notice, contact ANSYS, Inc.

Published in the U.S.A.

# **Table of Contents**

Using This Manual	xiii
1. What's In This Manual	xiii
2. The Contents of the Fluent Manuals	xiii
3. Where to Find the Files Used in the Tutorials	xv
4. How To Use This Manual	xv
4.1. For the Beginner	xv
4.2. For the Experienced User	xv
5. Typographical Conventions Used In This Manual	
1. Introduction to Using ANSYS Fluent in ANSYS Workbench: Fluid Flow and Heat Transfer in a	Mixing
Elbow	1
1.1. Introduction	1
1.2. Prerequisites	
1.3. Problem Description	2
1.4. Setup and Solution	
1.4.1. Preparation	
1.4.2. Creating a Fluent Fluid Flow Analysis System in ANSYS Workbench	
1.4.3. Creating the Geometry in ANSYS DesignModeler	
1.4.4. Meshing the Geometry in the ANSYS Meshing Application	
1.4.5. Setting Up the CFD Simulation in ANSYS Fluent	
1.4.5.1. Starting ANSYS Fluent	27
1.4.5.2. Setting Up Domain	
1.4.5.3. Setting Up Physics	
1.4.6. Solving	
1.4.7. Displaying Results in ANSYS Fluent and CFD-Post	
1.4.8. Duplicating the Fluent-Based Fluid Flow Analysis System	
1.4.9. Changing the Geometry in ANSYS DesignModeler	
1.4.10. Updating the Mesh in the ANSYS Meshing Application	
1.4.11. Calculating a New Solution in ANSYS Fluent	
1.4.12. Comparing the Results of Both Systems in CFD-Post	67
1.5. Summary	
2. Parametric Analysis in ANSYS Workbench Using ANSYS Fluent	
2.1. Introduction	
2.2. Prerequisites	
2.3. Problem Description	
2.4. Setup and Solution	
2.4.1. Preparation	
2.4.2. Adding Constraints to ANSYS DesignModeler Parameters in ANSYS Workbench	
2.4.3. Setting Up the CFD Simulation in ANSYS Fluent	
2.4.3.1. Starting ANSYS Fluent	
2.4.3.2. Setting Up Physics	
2.4.4. Defining Input Parameters in ANSYS Fluent	
2.4.5. Solving	
2.4.6. Postprocessing and Setting the Output Parameters in ANSYS CFD-Post	
2.4.7. Creating Additional Design Points in ANSYS Workbench	
2.4.8. Postprocessing the New Design Points in CFD-Post	
2.4.9. Summary	
3. Introduction to Using ANSYS Fluent: Fluid Flow and Heat Transfer in a Mixing Elbow	
3.1. Introduction	
3.2. Prerequisites	
3.3. Problem Description	122

3.4. Setup and Solution in Serial	122
3.4.1. Preparation	123
3.4.2. Launching ANSYS Fluent	123
3.4.3. Reading the Mesh	126
3.4.4. Setting Up Domain	
3.4.5. Setting Up Physics	
3.4.6. Solving	
3.4.7. Displaying the Preliminary Solution	
3.4.8. Using the Coupled Solver	
3.4.9. Adapting the Mesh	
3.5. Setup and Solution in Parallel	
3.5.1. Starting the Parallel Version of ANSYS Fluent	
3.5.1.1. Multiprocessor Machine	
3.5.1.2. Network of Computers	
•	
3.5.2. Reading and Partitioning the Mesh	
3.5.3. Solution	
5	
3.5.5. Postprocessing	
3.6. Summary	
4. Modeling Periodic Flow and Heat Transfer	
4.1. Introduction	
4.2. Prerequisites	
4.3. Problem Description	
4.4. Setup and Solution	
4.4.1. Preparation	
4.4.2. Mesh	
4.4.3. General Settings	
4.4.4. Models	
4.4.5. Materials	
4.4.6. Cell Zone Conditions	
4.4.7. Periodic Conditions	
4.4.8. Boundary Conditions	
4.4.9. Solution	
4.4.10. Postprocessing	
4.5. Summary	
4.6. Further Improvements	
5. Modeling External Compressible Flow	
5.1. Introduction	
5.2. Prerequisites	
5.3. Problem Description	
5.4. Setup and Solution	
5.4.1. Preparation	
5.4.2. Mesh	
5.4.3. Solver	
5.4.4. Models	
5.4.5. Materials	
5.4.6. Boundary Conditions	
5.4.7. Operating Conditions	
5.4.8. Solution	
5.4.9. Postprocessing	
5.5. Summary	
5.6. Further Improvements	265

6. Modeling Transient Compressible Flow	
6.1. Introduction	267
6.2. Prerequisites	267
6.3. Problem Description	268
6.4. Setup and Solution	268
6.4.1. Preparation	268
6.4.2. Reading and Checking the Mesh	269
6.4.3. Solver and Analysis Type	
6.4.4. Models	
6.4.5. Materials	273
6.4.6. Operating Conditions	
6.4.7. Boundary Conditions	
6.4.8. Solution: Steady Flow	
6.4.9. Enabling Time Dependence and Setting Transient Conditions	
6.4.10. Specifying Solution Parameters for Transient Flow and Solving	
6.4.11. Saving and Postprocessing Time-Dependent Data Sets	
6.5. Summary	
6.6. Further Improvements	
7. Modeling Radiation and Natural Convection	
7.1. Introduction	
7.2. Prerequisites	
7.3. Problem Description	
7.4. Setup and Solution	
7.4.1. Preparation	
7.4.2. Reading and Checking the Mesh	
7.4.3. Solver and Analysis Type	
7.4.4. Models	
7.4.5. Defining the Materials	
7.4.6. Operating Conditions	
7.4.7. Boundary Conditions	
•	
7.4.8. Obtaining the Solution	
7.4.9. Postprocessing	
7.4.10. Comparing the Contour Plots after Varying Radiating Surfaces	
7.4.11. S2S Definition, Solution, and Postprocessing with Partial Enclosure	
7.5. Summary	
7.6. Further Improvements	
8. Modeling Flow Through Porous Media	
8.1. Introduction	
8.2. Prerequisites	
8.3. Problem Description	
8.4. Setup and Solution	
8.4.1. Preparation	
8.4.2. Mesh	
8.4.3. General Settings	
8.4.4. Models	
8.4.5. Materials	
8.4.6. Cell Zone Conditions	
8.4.7. Boundary Conditions	
8.4.8. Solution	
8.4.9. Postprocessing	
8.5. Summary	
8.6. Further Improvements	392

9. Using a Single Rotating Reference Frame	393
9.1. Introduction	393
9.2. Prerequisites	393
9.3. Problem Description	394
9.4. Setup and Solution	395
9.4.1. Preparation	395
9.4.2. Mesh	396
9.4.3. General Settings	396
9.4.4. Models	398
9.4.5. Materials	400
9.4.6. Cell Zone Conditions	400
9.4.7. Boundary Conditions	401
9.4.8. Solution Using the Standard k- $\epsilon$ Model	404
9.4.9. Postprocessing for the Standard k- $\epsilon$ Solution	411
9.4.10. Solution Using the RNG k- $\epsilon$ Model	420
9.4.11. Postprocessing for the RNG k- $\epsilon$ Solution	422
9.5. Summary	
9.6. Further Improvements	427
9.7. References	428
10. Using Multiple Reference Frames	429
10.1. Introduction	429
10.2. Prerequisites	430
10.3. Problem Description	430
10.4. Setup and Solution	431
10.4.1. Preparation	431
10.4.2. Reading and Checking the Mesh and Setting the Units	432
10.4.3. Specifying Solver and Analysis Type	
10.4.4. Specifying the Models	
10.4.5. Specifying Materials	
10.4.6. Specifying Cell Zone Conditions	
10.4.7. Setting Boundary Conditions	438
10.4.8. Defining Mesh Interfaces	
10.4.9. Obtaining the Solution	
10.4.10. Step 9: Postprocessing	446
10.5. Summary	
10.6. Further Improvements	
11. Using Sliding Meshes	
11.1. Introduction	
11.2. Prerequisites	
11.3. Problem Description	
11.4. Setup and Solution	
11.4.1. Preparation	
11.4.2. Mesh	
11.4.3. General Settings	
11.4.4. Models	
11.4.5. Materials	
11.4.6. Cell Zone Conditions	
11.4.7. Boundary Conditions	
11.4.8. Operating Conditions	
11.4.9. Mesh Interfaces	
11.4.10. Solution	
11.4.11. Postprocessing	490

11.5. Summary	
11.6. Further Improvements	
12. Using Dynamic Meshes	
12.1. Introduction	
12.2. Prerequisites	
12.3. Problem Description	502
12.4. Setup and Solution	502
12.4.1. Preparation	
12.4.2. Mesh	
12.4.3. General Settings	
12.4.4. Models	
12.4.5. Materials	
12.4.6. Boundary Conditions	
12.4.7. Solution: Steady Flow	
12.4.8. Time-Dependent Solution Setup	
12.4.9. Mesh Motion	
12.4.10. Time-Dependent Solution	
12.4.11. Postprocessing	
12.5. Summary	535
12.6. Further Improvements	
13. Modeling Species Transport and Gaseous Combustion	
13.1. Introduction	
13.2. Prerequisites	
13.3. Problem Description	
13.4. Background	
13.5. Setup and Solution	
13.5.1. Preparation	
13.5.2. Mesh	
13.5.3. General Settings	
13.5.4. Models	
13.5.5. Materials	
13.5.6. Boundary Conditions	
13.5.7. Initial Reaction Solution	
13.5.8. Postprocessing	
13.5.9. NOx Prediction	
13.6. Summary	
13.7. Further Improvements	
14. Using the Non-Premixed Combustion Model	
14.1. Introduction	
14.2. Prerequisites	
14.3. Problem Description	
14.4. Setup and Solution	
14.4.1. Preparation	
14.4.2. Reading and Checking the Mesh	
14.4.3. Specifying Solver and Analysis Type	
14.4.4. Specifying the Models	
14.4.5. Defining Materials and Properties	
14.4.6. Specifying Boundary Conditions	
14.4.7. Specifying Operating Conditions	
14.4.8. Obtaining Solution	
14.4.9. Postprocessing	
14.4.10. Energy Balances Reporting	613

14.5. Summary	615
14.6. References	615
14.7. Further Improvements	616
15. Modeling Surface Chemistry	
15.1. Introduction	
15.2. Prerequisites	
15.3. Problem Description	
15.4. Setup and Solution	
15.4.1. Preparation	
15.4.2. Reading and Checking the Mesh	
15.4.3. Solver and Analysis Type	
15.4.4. Specifying the Models	
15.4.5. Defining Materials and Properties	
15.4.6. Specifying Boundary Conditions	
15.4.7. Setting the Operating Conditions	
15.4.8. Simulating Non-Reacting Flow	
15.4.9. Simulating Reacting Flow	
15.4.10. Postprocessing the Solution Results	
15.5. Summary	
15.6. Further Improvements	
16. Modeling Evaporating Liquid Spray	
16.1. Introduction	
16.2. Prerequisites	
16.3. Problem Description	
16.4. Setup and Solution	
16.4.1. Preparation	
16.4.2. Mesh	
16.4.3. Solver	
16.4.4. Models	
16.4.5. Materials	
16.4.6. Boundary Conditions	
16.4.7. Initial Solution Without Droplets	
16.4.8. Creating a Spray Injection	
5 1 7 7	
16.4.9. Solution	
16.4.10. Postprocessing	
16.5. Summary	
16.6. Further Improvements	
17. Using the VOF Model	
17.1. Introduction	
17.2. Prerequisites	
17.3. Problem Description	
17.4. Setup and Solution	
17.4.1. Preparation	
17.4.2. Reading and Manipulating the Mesh	
17.4.3. General Settings	
17.4.4. Models	
17.4.5. Materials	
17.4.6. Phases	
17.4.7. Operating Conditions	
17.4.8. User-Defined Function (UDF)	
17.4.9. Boundary Conditions	
17.4.10. Solution	735

17.4.11. Postprocessing	742
17.5. Summary	746
17.6. Further Improvements	746
18. Modeling Cavitation	747
18.1. Introduction	747
18.2. Prerequisites	747
18.3. Problem Description	747
18.4. Setup and Solution	
18.4.1. Preparation	
18.4.2. Reading and Checking the Mesh	
18.4.3. Solver Settings	
18.4.4. Models	
18.4.5. Materials	
18.4.6. Phases	
18.4.7. Boundary Conditions	
18.4.8. Operating Conditions	
18.4.9. Solution	
18.4.10. Postprocessing	
18.5. Summary	
18.6. Further Improvements	
19. Using the Mixture and Eulerian Multiphase Models	
19.1. Introduction	
19.2. Prerequisites	
19.3. Problem Description	
19.4. Setup and Solution	
19.4. 1. Preparation	
19.4.2. Mesh	
19.4.2. Mesh	
19.4.4. Models	
19.4.5. Materials	
19.4.6. Phases	
19.4.7. Boundary Conditions	
19.4.8. Operating Conditions	
19.4.9. Solution Using the Mixture Model	
19.4.10. Postprocessing for the Mixture Solution	
19.4.11. Higher Order Solution using the Mixture Model	
19.4.12. Setup and Solution for the Eulerian Model	
19.4.13. Postprocessing for the Eulerian Model	
19.5. Summary	
19.6. Further Improvements	
20. Modeling Solidification	
20.1. Introduction	
20.2. Prerequisites	
20.3. Problem Description	
20.4. Setup and Solution	
20.4.1. Preparation	
20.4.2. Reading and Checking the Mesh	
20.4.3. Specifying Solver and Analysis Type	
20.4.4. Specifying the Models	
20.4.5. Defining Materials	
20.4.6. Setting the Cell Zone Conditions	
20.4.7. Setting the Boundary Conditions	816

	20.4.8. Solution: Steady Conduction	824
	20.4.9. Solution: Transient Flow and Heat Transfer	833
	20.5. Summary	844
	20.6. Further Improvements	844
21.	Using the Eulerian Granular Multiphase Model with Heat Transfer	845
	21.1. Introduction	845
	21.2. Prerequisites	845
	21.3. Problem Description	846
	21.4. Setup and Solution	846
	21.4.1. Preparation	847
	21.4.2. Mesh	848
	21.4.3. Solver Settings	849
	21.4.4. Models	849
	21.4.5. UDF	850
	21.4.6. Materials	851
	21.4.7. Phases	852
	21.4.8. Boundary Conditions	855
	21.4.9. Solution	862
	21.4.10. Postprocessing	874
	21.5. Summary	877
	21.6. Further Improvements	877
	21.7. References	877
22.	Postprocessing	879
	22.1. Introduction	879
	22.2. Prerequisites	880
	22.3. Problem Description	880
	22.4. Setup and Solution	880
	22.4.1. Preparation	881
	22.4.2. Reading the Mesh	882
	22.4.3. Manipulating the Mesh in the Viewer	882
	22.4.4. Adding Lights	884
	22.4.5. Creating Isosurfaces	889
	22.4.6. Generating Contours	891
	22.4.7. Generating Velocity Vectors	896
	22.4.8. Creating an Animation	902
	22.4.9. Displaying Pathlines	907
	22.4.10. Creating a Scene With Vectors and Contours	914
	22.4.11. Advanced Overlay of Pathlines on a Scene	915
	22.4.12. Creating Exploded Views	917
	22.4.13. Animating the Display of Results in Successive Streamwise Planes	922
	22.4.14. Generating XY Plots	924
	22.4.15. Creating Annotation	928
	22.4.16. Saving Picture Files	930
	22.4.17. Generating Volume Integral Reports	
	22.5. Summary	
23	Using the Adjoint Solver – 2D Laminar Flow Past a Cylinder	
	23.1. Introduction	
	23.2. Prerequisites	
	23.3. Problem Description	
	23.4. Setup and Solution	
	23.4.1. Preparation	
	23.4.2. Step 1: Load the Adjoint Solver Add-on	937

23.4.3. Step 2: Define Observables	
23.4.4. Step 3: Compute the Drag Sensitivity	
23.4.5. Step 4: Postprocess and Export Drag Sensitivity	
23.4.5.1. Boundary Condition Sensitivity	
23.4.5.2. Momentum Source Sensitivity	
23.4.5.3. Shape Sensitivity	
23.4.5.4. Exporting Drag Sensitivity Data	950
23.4.6. Step 5: Compute Lift Sensitivity	952
23.4.7. Step 6: Modify the Shape	953
23.5. Summary	959
24. Simulating a Single Battery Cell Using the MSMD Battery Model	961
24.1. Introduction	
24.2. Prerequisites	
24.3. Problem Description	
24.4. Setup and Solution	
24.4.1. Preparation	
24.4.2. Reading and Scaling the Mesh	
24.4.3. Loading the MSMD battery Add-on	
24.4.4. NTGK Battery Model Setup	
24.4.4.1. Specifying Solver and Models	
24.4.4.2. Defining New Materials for Cell and Tabs	
24.4.4.3. Defining Cell Zone Conditions	
24.4.4.4. Defining Boundary Conditions	
24.4.4.5. Specifying Solution Settings	
24.4.4.6. Obtaining Solution	
24.4.5. Postprocessing	
24.4.6. Simulating the Battery Pulse Discharge Using the ECM Model	
24.4.7. Using the Reduced Order Method (ROM)	992
24.4.8. External and Internal Short-Circuit Treatment	993
24.4.8.1. Setting up and Solving a Short-Circuit Problem	993
24.4.8.2. Postprocessing	
24.5. Summary	
24.6. Appendix	1001
24.7. References	1003
25. Simulating a 1P3S Battery Pack Using the MSMD Battery Model	1005
25.1. Introduction	
25.2. Prerequisites	1005
25.3. Problem Description	1006
25.4. Setup and Solution	1006
25.4.1. Preparation	1006
25.4.2. Reading and Scaling the Mesh	
25.4.3. Loading the MSMD battery Add-on	
25.4.4. Battery Model Setup	1009
25.4.4.1. Specifying Solver and Models	1009
25.4.4.2. Defining New Materials	
25.4.4.3. Defining Cell Zone Conditions	
25.4.4.4. Defining Boundary Conditions	
25.4.4.5. Specifying Solution Settings	
25.4.4.6. Obtaining Solution	
25.4.5. Postprocessing	
25.5. Summary	1033

# **Using This Manual**

This preface is divided into the following sections:

- 1. What's In This Manual
- 2. The Contents of the Fluent Manuals
- 3. Where to Find the Files Used in the Tutorials
- 4. How To Use This Manual
- 5. Typographical Conventions Used In This Manual

# 1. What's In This Manual

The ANSYS Fluent Tutorial Guide contains a number of tutorials that teach you how to use ANSYS Fluent to solve different types of problems. In each tutorial, features related to problem setup and postprocessing are demonstrated.

The tutorials are written with the assumption that you have completed one or more of the introductory tutorials found in this manual:

- Introduction to Using ANSYS Fluent in ANSYS Workbench: Fluid Flow and Heat Transfer in a Mixing Elbow (p. 1)
- Parametric Analysis in ANSYS Workbench Using ANSYS Fluent (p. 73)
- Introduction to Using ANSYS Fluent: Fluid Flow and Heat Transfer in a Mixing Elbow (p. 121)

and that you are familiar with the ANSYS Fluent navigation pane and ribbon structure. Some steps in the setup and solution procedure will not be shown explicitly.

Introduction to Using ANSYS Fluent: Fluid Flow and Heat Transfer in a Mixing Elbow (p. 121)

All of the tutorials include some postprocessing instructions, but Postprocessing (p. 879) is devoted entirely to postprocessing.

# 2. The Contents of the Fluent Manuals

The manuals listed below form the Fluent product documentation set. They include descriptions of the procedures, commands, and theoretical details needed to use Fluent products. These manuals are all available on the ANSYS Customer Portal (http://support.ansys.com/documentation).

- Fluent Getting Started Guide contains general information about getting started with using Fluent and provides details about starting, running, and exiting the program.
- Fluent Migration Manual contains information about transitioning from the previous release of Fluent, including details about new features, solution changes, and text command list changes.
- Fluent User's Guide contains detailed information about running a simulation using the solution mode of Fluent, including information about the user interface, reading and writing files, defining boundary conditions, setting up physical models, calculating a solution, and analyzing your results.
- Fluent Meshing User's Guide contains detailed information about creating 3D meshes using the meshing mode of Fluent.

- Fluent in Workbench User's Guide contains information about getting started with and using Fluent within the Workbench environment.
- Fluent Theory Guide contains reference information for how the physical models are implemented in Fluent.
- Fluent Customization Manual contains information about writing and using user-defined functions (UDFs).
- Fluent Tutorial Guide contains a number of examples of various flow problems with detailed instructions, commentary, and postprocessing of results.

Tutorials for release 18.0 are available on the ANSYS Customer Portal. To access tutorials and their input files on the ANSYS Customer Portal, go to http://support.ansys.com/training.

• ANSYS Fluent Meshing Tutorial Guide contains a number of examples of general mesh-generation techniques used in ANSYS Fluent Meshing.

Tutorials for release 18.0 are available on the ANSYS Customer Portal. To access tutorials and their input files on the ANSYS Customer Portal, go to http://support.ansys.com/training.

- Fluent Text Command List contains a brief description of each of the commands in Fluent's solution mode text interface.
- Fluent Meshing Text Command List contains a brief description of each of the commands in Fluent's meshing mode text interface.
- Fluent Advanced Add-On Modules contains information about the usage of the different advanced Fluent add-on modules, which are applicable for specific modeling needs.
  - Part I: ANSYS Fluent Adjoint Solver contains information about the background and usage of Fluent's Adjoint Solver Module that allows you to obtain detailed sensitivity data for the performance of a fluid system.
  - Part II: ANSYS Fluent Battery Module contains information about the background and usage of Fluent's Battery Module that allows you to analyze the behavior of electric batteries.
  - Part III: ANSYS Fluent Continuous Fiber Module contains information about the background and usage of Fluent's Continuous Fiber Module that allows you to analyze the behavior of fiber flow, fiber properties, and coupling between fibers and the surrounding fluid due to the strong interaction that exists between the fibers and the surrounding gas.
  - Part IV: ANSYS Fluent Fuel Cell Modules contains information about the background and the usage of two separate add-on fuel cell models for Fluent that allow you to model polymer electrolyte membrane fuel cells (PEMFC), solid oxide fuel cells (SOFC), and electrolysis with Fluent.
  - Part V: ANSYS Fluent Magnetohydrodynamics (MHD) Module contains information about the background and usage of Fluent's Magnetohydrodynamics (MHD) Module that allows you to analyze the behavior of electrically conducting fluid flow under the influence of constant (DC) or oscillating (AC) electromagnetic fields.
  - Part VI: ANSYS Fluent Population Balance Module contains information about the background and usage of Fluent's Population Balance Module that allows you to analyze multiphase flows involving size distributions where particle population (as well as momentum, mass, and energy) require a balance equation.

- Fluent as a Server User's Guide contains information about the usage of Fluent as a Server which allows you to connect to a Fluent session and issue commands from a remote client application.
- Running ANSYS Fluent Using a Load Manager contains information about using third-party load managers with ANSYS Fluent.
  - Part I: Running ANSYS Fluent Under LSF contains information about using Fluent with Platform Computing's LSF software, a distributed computing resource management tool.
  - Part II: Running ANSYS Fluent Under PBS Professional contains information about using Fluent with Altair PBS Professional, an open workload management tool for local and distributed environments.
  - Part III: Running ANSYS Fluent Under SGE contains information about using Fluent with Univa Grid Engine (formerly Sun Grid Engine) software, a distributed computing resource management tool.

# 3. Where to Find the Files Used in the Tutorials

Each of the tutorials uses an existing mesh file. (Tutorials for mesh generation are provided with the mesh generator documentation.) You will find the appropriate mesh file (and any other relevant files used in the tutorial) on the ANSYS Customer Portal. The "Preparation" step of each tutorial will tell you where to find the necessary files. (Note that Tutorial Postprocessing (p. 879) use existing case and data files.)

## 4. How To Use This Manual

Depending on your familiarity with computational fluid dynamics and the ANSYS Fluent software, you can use this tutorial guide in a variety of ways.

### 4.1. For the Beginner

If you are a beginning user of ANSYS Fluent you should first read and solve Tutorial 1, in order to familiarize yourself with the interface and with basic setup and solution procedures. You may then want to try a tutorial that demonstrates features that you are going to use in your application. For example, if you are planning to solve a problem using the non-premixed combustion model, you should look at Using the Non-Premixed Combustion Model (p. 583).

You may want to refer to other tutorials for instructions on using specific features, such as custom field functions, mesh scaling, and so on, even if the problem solved in the tutorial is not of particular interest to you. To learn about postprocessing, you can look at Postprocessing (p. 879), which is devoted entirely to postprocessing (although the other tutorials all contain some postprocessing as well).

### 4.2. For the Experienced User

If you are an experienced ANSYS Fluent user, you can read and/or solve the tutorial(s) that demonstrate features that you are going to use in your application. For example, if you are planning to solve a problem using the non-premixed combustion model, you should look at Using the Non-Premixed Combustion Model (p. 583).

You may want to refer to other tutorials for instructions on using specific features, such as custom field functions, mesh scaling, and so on, even if the problem solved in the tutorial is not of particular interest to you. To learn about postprocessing, you can look at Postprocessing (p. 879), which is devoted entirely to postprocessing (although the other tutorials all contain some postprocessing as well).

# **5. Typographical Conventions Used In This Manual**

Several typographical conventions are used in this manual's text to help you find commands in the user interface.

• Different type styles are used to indicate graphical user interface items and text interface items. For example:

```
Iso-Surface dialog box
surface/iso-surface text command
```

• The text interface type style is also used when illustrating exactly what appears on the screen to distinguish it from the narrative text. In this context, user inputs are typically shown in boldface. For example,

```
solve/initialize/set-fmg-initialization
```

```
Customize your FMG initialization:
set the number of multigrid levels [5]
set FMG parameters on levels ..
residual reduction on level 1 is: [0.001]
number of cycles on level 1 is: [10] 100
residual reduction on level 2 is: [0.001]
number of cycles on level 2 is: [50] 100
```

• Mini flow charts are used to guide you through the ribbon or the tree, leading you to a specific option, dialog box, or task page. The following tables list the meaning of each symbol in the mini flow charts.

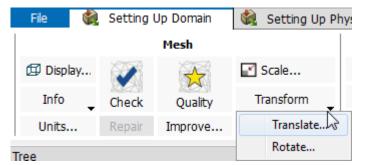
**Table 1: Mini Flow Chart Symbol Descriptions** 

Symbol	Indicated Action
10000 Million 10000	Look at the ribbon
F=_	Look at the tree
¢	Double-click to open task page
	Select from task page
Ċ	Right-click the preceding item

For example,

Setting Up Domain  $\rightarrow$  Mesh  $\rightarrow$  Transform  $\rightarrow$  Translate...

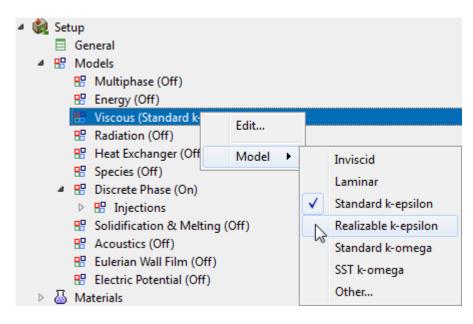
indicates selecting the **Setting Up Domain** ribbon tab, clicking **Transform** (in the **Mesh** group box) and selecting **Translate...**, as indicated in the figure below:



And

# **For a setup** $\rightarrow$ Models $\rightarrow$ Viscous $\stackrel{\mu}{\hookrightarrow}$ Model $\rightarrow$ Realizable k-epsilon

indicates expanding the **Setup** and **Models** branches, right-clicking **Viscous**, and selecting **Realizable k-epsilon** from the **Model** sub-menu, as shown in the following figure:



And

# **E** Setup $\rightarrow$ **Conditions** $\rightarrow$ **E** porous-in

indicates opening the task page as shown below:

<ul> <li>Setup         <ul> <li>General</li> <li>Models</li> <li>Materials</li> <li>Cell Zone Conditions</li> </ul> </li> <li>Cell Zone Conditions         <ul> <li>Cell Zone Conditions</li> <li>Cell Zone Conditions</li> <li>Cell autors</li> <li>default-interior (interior)</li> <li>default-interior:010 (interior)</li> <li>cell to efault-interior:010 (interior)</li> <li>cell to efault-interior:010 (interior)</li> <li>context (pressure-outlet)</li> <li>porous-in (interior)</li> <li>porous-out (interior)</li> <li>substrate-wall (wall)</li> <li>wall (wall)</li> <li>Dynamic Mesh</li> <li>Reference Values</li> </ul> </li> <li>Methods         <ul> <li>Controls</li> </ul> </li> </ul>	Boundary Conditions Zone Filter Text default-interior default-interior:010 inlet outlet porous-out substrate-wall wall
<ul> <li>Report Definitions</li> <li>Monitors</li> <li>Cell Registers</li> <li>Initialization</li> <li>Calculation Activities</li> <li>Run Calculation</li> <li>Results</li> <li>Graphics</li> <li>Plots</li> <li>Scene</li> <li>Animations</li> </ul>	Phase       Type       ID         mixture       interior       6         Edit       Copy       Profiles         Parameters       Operating Conditions         Display Mesh       Periodic Conditions         Highlight Zone       Help

In this manual, mini flow charts usually accompany a description of a dialog box or command, or a screen illustration showing how to use the dialog box or command. They show you how to quickly access a command or dialog box without having to search the surrounding material.

In-text references to File ribbon tab selections can be indicated using a "/". For example File/Write/Case...
indicates clicking the File ribbon tab and selecting Case... from the Write submenu (which opens the Select
File dialog box).

# Chapter 1: Introduction to Using ANSYS Fluent in ANSYS Workbench: Fluid Flow and Heat Transfer in a Mixing Elbow

This tutorial is divided into the following sections:

- 1.1. Introduction
- 1.2. Prerequisites
- 1.3. Problem Description
- 1.4. Setup and Solution
- 1.5. Summary

## **1.1.Introduction**

This tutorial illustrates using ANSYS Fluent fluid flow systems in ANSYS Workbench to set up and solve a three-dimensional turbulent fluid-flow and heat-transfer problem in a mixing elbow. It is designed to introduce you to the ANSYS Workbench tool set using a simple geometry. Guided by the steps that follow, you will create the elbow geometry and the corresponding computational mesh using the geometry and meshing tools within ANSYS Workbench. You will use ANSYS Fluent to set up and solve the CFD problem, then visualize the results in both ANSYS Fluent and in the CFD-Post postprocessing tool. Some capabilities of ANSYS Workbench (for example, duplicating fluid flow systems, connecting systems, and comparing multiple data sets) are also examined in this tutorial.

This tutorial demonstrates how to do the following:

- Launch ANSYS Workbench.
- Create a Fluent fluid flow analysis system in ANSYS Workbench.
- · Create the elbow geometry using ANSYS DesignModeler.
- Create the computational mesh for the geometry using ANSYS Meshing.
- Set up the CFD simulation in ANSYS Fluent, which includes:
  - Setting material properties and boundary conditions for a turbulent forced-convection problem.
  - Initiating the calculation with residual plotting.
  - Calculating a solution using the pressure-based solver.
  - Examining the flow and temperature fields using ANSYS Fluent and CFD-Post.
- Create a copy of the original Fluent fluid flow analysis system in ANSYS Workbench.
- Change the geometry in ANSYS DesignModeler, using the duplicated system.
- Regenerate the computational mesh.
- Recalculate a solution in ANSYS Fluent.

Introduction to Using ANSYS Fluent in ANSYS Workbench: Fluid Flow and Heat Transfer in a Mixing Elbow

· Compare the results of the two calculations in CFD-Post.

## **1.2. Prerequisites**

This tutorial assumes that you have little to no experience with ANSYS Workbench, ANSYS DesignModeler, ANSYS Meshing, ANSYS Fluent, or CFD-Post, and so each step will be explicitly described.

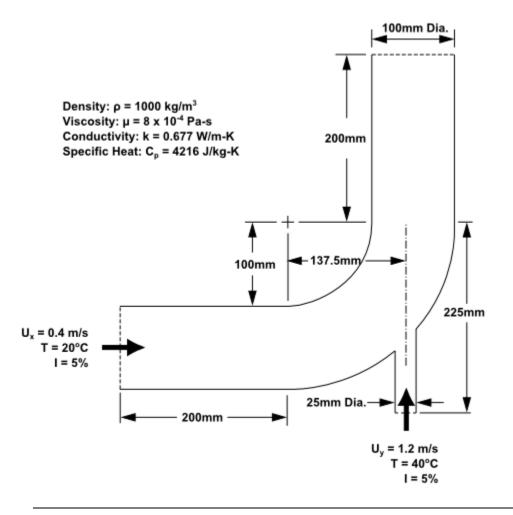
# **1.3. Problem Description**

The problem to be considered is shown schematically in Figure 1.1: Problem Specification (p. 3). A cold fluid at 293.15 K flows into the pipe through a large inlet and mixes with a warmer fluid at 313.15 K that enters through a smaller inlet located at the elbow. The mixing elbow configuration is encountered in piping systems in power plants and process industries. It is often important to predict the flow field and temperature field in the area of the mixing region in order to properly design the junction.

#### Note

Because the geometry of the mixing elbow is symmetric, only half of the elbow must be modeled.

#### **Figure 1.1: Problem Specification**



#### Note

The functionality to create named selections exists in both ANSYS DesignModeler and ANSYS Meshing. For the purposes of this tutorial, named selections are created in ANSYS Meshing since the meshing application provides more comprehensive and extensive named selection functionality.

## 1.4. Setup and Solution

To help you quickly identify graphical user interface items at a glance and guide you through the steps of setting up and running your simulation, the ANSYS Fluent Tutorial Guide uses several type styles and mini flow charts. See Typographical Conventions Used In This Manual (p. xvi) for detailed information.

The following sections describe the setup and solution steps for this tutorial:

1.4.1. Preparation

- 1.4.2. Creating a Fluent Fluid Flow Analysis System in ANSYS Workbench
- 1.4.3. Creating the Geometry in ANSYS DesignModeler
- 1.4.4. Meshing the Geometry in the ANSYS Meshing Application
- 1.4.5. Setting Up the CFD Simulation in ANSYS Fluent

1.4.6. Solving

- 1.4.7. Displaying Results in ANSYS Fluent and CFD-Post
- 1.4.8. Duplicating the Fluent-Based Fluid Flow Analysis System

Introduction to Using ANSYS Fluent in ANSYS Workbench: Fluid Flow and Heat Transfer in a Mixing Elbow

1.4.9. Changing the Geometry in ANSYS DesignModeler

1.4.10. Updating the Mesh in the ANSYS Meshing Application

1.4.11. Calculating a New Solution in ANSYS Fluent

1.4.12. Comparing the Results of Both Systems in CFD-Post

### 1.4.1. Preparation

- 1. Set up a working folder on the computer you will be using.
- 2. Go to the ANSYS Customer Portal, https://support.ansys.com/training.

#### Note

If you do not have a login, you can request one by clicking **Customer Registration** on the log in page.

- 3. Enter the name of this tutorial into the search bar.
- 4. Narrow the results by using the filter on the left side of the page.
  - a. Click ANSYS Fluent under Product.
  - b. Click 18.0 under Version.
- 5. Select this tutorial from the list.
- 6. Click the **elbow-workbench\_R180.zip** link to download the input files.
- 7. Unzip elbow-workbench\_R180.zip to your working folder. This file contains a folder, elbowworkbench, that holds the following items:
  - two geometry files, elbow\_geometry.agdb and elbow\_geometry.stp
  - an ANSYS Workbench project archive, elbow-workbench.wbpz

#### Тір

The Workbench project archive contains the project as it will be once you have completed all of the steps of the tutorial and is included for reference. If you want to extract the project archive, start Workbench and select the **File**  $\rightarrow$  **Restore Archive...** menu item. You will be prompted with a dialog box to specify a location in which to extract the project and its supporting files. You may choose any convenient location.

#### Note

ANSYS Fluent tutorials are prepared using ANSYS Fluent on a Windows system. The screen shots and graphic images in the tutorials may be slightly different than the appearance on your system, depending on the operating system or graphics card.

## 1.4.2. Creating a Fluent Fluid Flow Analysis System in ANSYS Workbench

In this step, you will start ANSYS Workbench, create a new Fluent fluid flow analysis system, then review the list of files generated by ANSYS Workbench.

1. From the Windows Start menu, select Start > All Programs > ANSYS 18.0 > Workbench 18.0 to start a new ANSYS Workbench session.

This displays the ANSYS Workbench application window, which has the **Toolbox** on the left and the **Project Schematic** to its right. Various supported applications are listed in the **Toolbox** and the components of the analysis system will be displayed in the **Project Schematic**.

#### Note

Depending on which other products you have installed, the analysis systems that appear may differ from those in the figures that follow in this tutorial.

2. Create a new Fluent fluid flow analysis system by double-clicking the **Fluid Flow (Fluent)** option under **Analysis Systems** in the **Toolbox**.

#### Tip

You can also drag-and-drop the analysis system into the **Project Schematic**. A green dotted outline indicating a potential location for the new system initially appears in the **Project Schematic**. When you drag the system to one of the outlines, it turns into a red box to indicate the chosen location of the new system.

🔥 Unsaved Project - Workbench	
<u>File View Tools U</u> nits Ex	tensions Jobs <u>H</u> elp
Project	
👔 Import 🗟 🖗 Reconnect 👔 R	Refresh Project 🛛 🖉 Update Project 🛛 📲 ACT Start Page
Toolbox	▼ ₽ X         Project Schematic         ▼ ₽ X
Analysis Systems	
🔄 Fluid Flow (Fluent)	
IC Engine (Fluent)	
Throughflow	Fluid Flow analysis using FLUENT solver
Throughflow (BladeGen)	
External Connection Systems     ■	
View All / Cu	istomize
Ready	Job Monitor

🔥 Unsaved Project - Workbench						
File View Tools Units Extensions Jobs Help						
Project	Project					
👔 Import 🗟 Reconnect 👔 Refresh Project 🍼 Update Project 📲 ACT Start Page						
Toolbox 🝷 🕂 🗙	Toolbox 👻 👎 🗶 Project Schematic					
Analysis Systems						
S Fluid Flow (Fluent)						
🞽 IC Engine (Fluent)	▼ A					
🚽 Throughflow	1 💽 Fluid Flow (Fluent)					
☐ Throughflow (BladeGen)	2 🥪 Geometry 🛛 🤶 🖌					
	3 🎯 Mesh 🔗 🖌					
	4 🍓 Setup 🔗 🚽					
	5 🕼 Solution 🔗 🖌					
	6 😡 Results 🛛 😨 🛓					
	Fluid Flow (Fluent)					

Figure 1.3: ANSYS Workbench with a New Fluent-Based Fluid Flow Analysis System

- 3. Name the analysis.
  - a. Double-click the Fluid Flow (Fluent) label underneath the analysis system (if it is not already highlighted).
  - b. Enter elbow for the name of the analysis system.
- 4. Save the project.
  - a. In ANSYS Workbench, under the File menu, select Save.

#### $\textbf{File} \rightarrow \textbf{Save}$

This displays the **Save As** dialog box, where you can browse to your working folder and enter a specific name for the ANSYS Workbench project.

b. In your working directory, enter elbow-workbench as the project **File name** and click the **Save** button to save the project. ANSYS Workbench saves the project with a .wbpj extension and also saves supporting files for the project.

Note that the fluid flow analysis system is composed of various *cells* (**Geometry**, **Mesh**, etc.) that represent the workflow for performing the analysis. ANSYS Workbench is composed of multiple data-integrated and native applications in a single, seamless project flow, where individual cells can obtain data from other cells and provide data to other cells. As a result of this constant flow of data, a cell's state can quickly change. ANSYS Workbench provides a visual indication of a cell's state at any given time via icons on the right side of each cell. Brief descriptions of the various states are provided below:

• Unfulfilled (<sup>\*</sup>) indicates that required upstream data does not exist. For example, when you first create a new Fluid Flow (Fluent) analysis system, all cells downstream of the Geometry cell appear as Unfulfilled because you have not yet specified a geometry for the system.

Introduction to Using ANSYS Fluent in ANSYS Workbench: Fluid Flow and Heat Transfer in a Mixing Elbow

- **Refresh Required** (*₹*) indicates that upstream data has changed since the last refresh or update. For example, after you assign a geometry to the geometry cell in your new **Fluid Flow (Fluent)** analysis system, the **Mesh** cell appears as **Refresh Required** since the geometry data has not yet been passed from the **Geometry** cell to the **Mesh** cell.
- Attention Required (?) indicates that the current upstream data has been passed to the cell, however, you must take some action to proceed. For example, after you launch ANSYS Fluent from the Setup cell in a Fluid Flow (Fluent) analysis system that has a valid mesh, the Setup cell appears as Attention Required because additional data must be entered in ANSYS Fluent before you can calculate a solution.
- Update Required (<sup>≁</sup>) indicates that local data has changed and the output of the cell must be regenerated. For example, after you launch ANSYS Meshing from the Mesh cell in a Fluid Flow (Fluent) analysis system that has a valid geometry, the Mesh cell appears as Update Required because the Mesh cell has all the data it must generate an ANSYS Fluent mesh file, but the ANSYS Fluent mesh file has not yet been generated.
- Up To Date (
   ) indicates that an update has been performed on the cell and no failures have occurred or that an interactive calculation has been completed successfully. For example, after ANSYS Fluent finishes performing the number of iterations that you request, the Solution cell appears as Upto-Date.
- Interrupted, Update Required (<sup>\*</sup>) indicates that you have interrupted an update (or canceled an interactive calculation that is in progress). For example, if you select the **Cancel** button in ANSYS Fluent while it is iterating, ANSYS Fluent completes the current iteration and then the **Solution** cell appears as **Interrupted**, **Update Required**.
- Input Changes Pending (
   ) indicates that the cell is locally up-to-date, but may change when
  next updated as a result of changes made to upstream cells. For example, if you change the Mesh
  in an Up-to-Date Fluid Flow (Fluent) analysis system, the Setup cell appears as Refresh Required,
  and the Solution and Results cells appear as Input Changes Pending.
- **Pending** (<sup>\*</sup>) indicates that a batch or asynchronous solution is in progress. When a cell enters the **Pending** state, you can interact with the project to exit Workbench or work with other parts of the project. If you make changes to the project that are upstream of the updating cell, then the cell will not be in an up-to-date state when the solution completes.

For more information about cell states, see the Workbench User's Guide.

5. View the list of files generated by ANSYS Workbench.

ANSYS Workbench allows you to easily view the files associated with your project using the **Files** view. To open the **Files** view, select the **Files** option under the **View** menu at the top of the ANSYS Workbench window.

 $View \rightarrow Files$ 

# Figure 1.4: ANSYS Workbench Files View for the Project After Adding a Fluent-Based Fluid Flow Analysis System

Files (						
	А	В	С	D	E	
1	Name 💌	Cell 🔽 ID	Size 💌	Туре 💌	Date Modified 💌	Location
2	🔥 elbow-workbench.wbpj		123 KB	ANSYS Project File	XX/XX/20XX 0:00:00 AM	C:\Tutorial_01
3	🂐 designPoint.wbdp		25 KB	Design Point File	XX/XX/20XX 0:00:00 AM	C:\Tutorial_01\elbow-wo

In the **Files** view, you will be able to see the name and type of file, the ID of the cell that the file is associated with, the size of the file, the location of the file, and other information. For more information about the **Files** view, see the Workbench User's Guide.

#### Note

The sizes of the files listed may differ slightly from those portrayed in Figure 1.4: ANSYS Workbench Files View for the Project After Adding a Fluent-Based Fluid Flow Analysis System (p. 9).

From here, you will create the geometry described in Figure 1.1: Problem Specification (p. 3), and later create a mesh and set up a fluid flow analysis for the geometry.

### 1.4.3. Creating the Geometry in ANSYS DesignModeler

For the geometry of your fluid flow analysis, you can create a geometry in ANSYS DesignModeler, ANSYS SpaceClaim Direct Modeler, or import the appropriate geometry file. In this step, you will create the geometry in ANSYS DesignModeler, then review the list of files generated by ANSYS Workbench.

#### Important

Note the **Attention Required** icon (?) within the **Geometry** cell for the system. This indicates that the cell requires data (for example, a geometry). Once the geometry is defined, the state of the cell will change accordingly. Likewise, the state of some of the remaining cells in the system will change.

#### Note

If you would rather not create the geometry in ANSYS DesignModeler, you can import a preexisting geometry by right-clicking the **Geometry** cell and selecting the **Import Geometry** option from the context menu. From there, you can browse your file system to locate the elbow\_geometry.agdb geometry file that is provided for this tutorial. If you do not have access to ANSYS DesignModeler, you can use the elbow\_geometry.stp file instead.

To learn how to create a mesh from the geometry you imported, go to Meshing the Geometry in the ANSYS Meshing Application (p. 20).

1. Start ANSYS DesignModeler.

In the ANSYS Workbench **Project Schematic**, right-click the **Geometry** cell in the elbow fluid flow analysis system to display the context menu, then select **New DesignModeler Geometry...**. This displays the ANSYS DesignModeler application.

2. Set the units in ANSYS DesignModeler.

When ANSYS DesignModeler first appears, you should select desired system of length units to work from. For the purposes of this tutorial (where you will create the geometry in millimeters and perform the CFD analysis using SI units) set the units to **Millimeter**.

#### Units → Millimeter

3. Create the geometry.

The geometry for this tutorial (Figure 1.1: Problem Specification (p. 3)) consists of a large curved pipe accompanied by a smaller side pipe. ANSYS DesignModeler provides various geometry primitives that can be combined to rapidly create geometries such as this one. You will perform the following tasks to create the geometry:

- Create the bend in the main pipe by defining a segment of a torus.
- Extrude the faces of the torus segment to form the straight inlet and outlet lengths.
- · Create the side pipe by adding a cylinder primitive.
- Use the symmetry tool to reduce the model to half of the pipe assembly, thus reducing computational cost.
- a. Create the main pipe:
  - i. Create a new torus for the pipe bend by choosing the **Create**  $\rightarrow$  **Primitives**  $\rightarrow$  **Torus** menu item from the menubar.

A preview of the torus geometry will appear in the graphics window. Note that this is a preview and the geometry has not been created yet. First you must specify the parameters of the torus primitive in the next step.

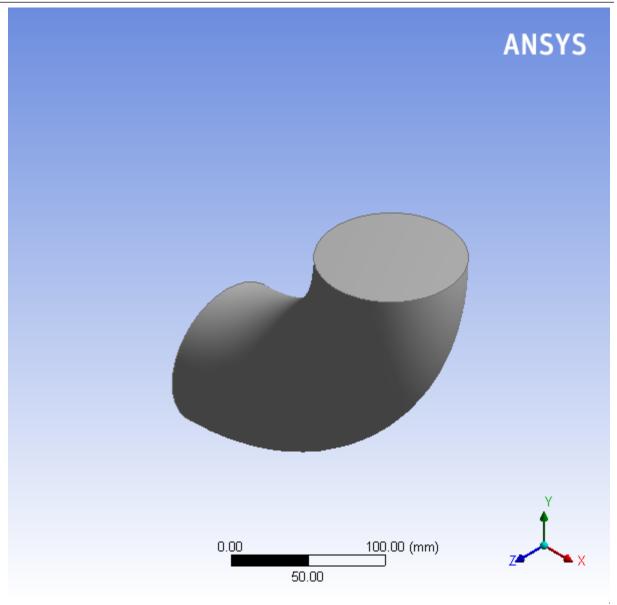
ii. In the **Details View** for the new torus (**Torus1**), set **Base Y Component** to -1 by clicking the **1** to the right of **FD10**, **Base Y Component**, entering -1, and pressing **Enter**. This specifies the direction vector from the origin to the center of the circular cross-section at the start of the torus. In the same manner, specify **Angle**; **Inner Radius**; and **Outer Radius** as shown below.

#### Note

Enter only the value without the units of mm. They will be appended automatically because you specified the units previously.

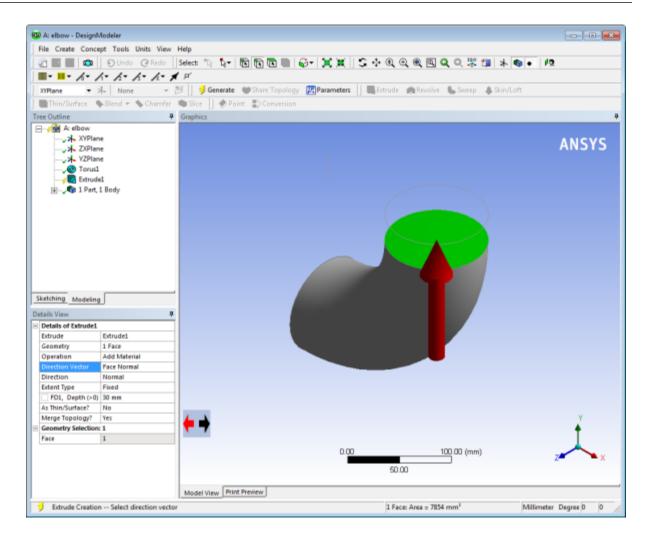
Details of Torus1	
Torus	Torus1
Base Plane	XYPlane
Operation	Add Material
Origin Definition	Coordinates
FD3, Origin X Coordinate	0 mm
FD4, Origin Y Coordinate	0 mm
FD5, Origin Z Coordinate	0 mm
Axis Definition	Components
FD6, Axis X Component	0
FD7, Axis Y Component	0
FD8, Axis Z Component	1
Base Definition	Components
FD9, Base X Component	0
FD10, Base Y Component	-1
FD11, Base Z Component	0
FD12, Angle (>0)	90 °
FD13, Inner Radius (>0)	100 mm
FD14, Outer Radius (>0)	200 mm
As Thin/Surface?	No

iii. To create the torus segment, click the **Generate** button <sup>Generate</sup> that is located in the ANSYS DesignModeler toolbar.



The Torus1 item appears in the **Tree Outline** view. If you want to delete this item, you can right-click it and select **Delete** from the context menu that opens.

- iv. Ensure that the selection filter is set to **Faces**. This is indicated by the **Faces** button **(b)** appearing depressed in the toolbar and the appearance of the Face selection cursor, **(b)** when you mouse over the geometry.
- v. Select the top face (in the positive Y direction) of the elbow and click the **Extrude** button from the **3D Features** toolbar.
- vi. In the **Details View** for the new extrusion (**Extrude1**), click **Apply** to the right of **Geometry**. This accepts the face you selected as the base geometry of the extrusion.
- vii. Click **None (Normal)** to the right of **Direction Vector**. Again, ensure that the selection filter is set to **Faces**, select the same face on the elbow to specify that the extrusion will be normal to the face and click **Apply**.



viii.Enter 200 for **FD1**, **Depth (>0)** and click **Generate**.

Introduction to Using ANSYS Fluent in ANSYS Workbench: Fluid Flow and Heat Transfer in a Mixing Elbow

😨 A: elbow - DesignN	Andeler		
	pt Tools View Help		
×			
		- 10 10 10 10 0-   I   I   S ∻ Q Q Q Q Q	<b>()</b> • 19
	K= K= K= K= # F		
XVPlane • 3	🐛 None 👻 🎽 🚽 Gener	ate 🐨 Share Topology 📆 Parameters	
Extrude 💏 Rev	olve 🐁 Sweep 🚷 Skin/Loft 🛛 🛅 Ti	nin/Surface 💊 Blend 🔻 💊 Chamfer 🏟 Slice 🛛 🚸 Point 📳 Conversion	
Tree Outline	a	Graphics	7
- 🖓 A: elbow			
	e		ANSYS
ZXPlan	e		ANSTS
	e		
, 🕑 Torus1			
Extrude			
E-Add that's	1 Body		
Sketching Modeling	1		
Details View	0		
<ul> <li>Details of Extrude1</li> </ul>			
Extrude	Extrude1		
Geometry	1 Face		
Operation Direction Vector	Add Material Face Normal		
Direction Vector	Normal		
Extent Type	Fixed		
FD1, Depth (>0)			
As Thin/Surface?	No		
Merge Topology?	Yes		
Geometry Selection			
Face	1		
			Y
			+
		0.00 100.00 (mm)	
			Z X
		50.00	
		Model View Print Preview	
Ready		No Selection	Millimeter 0 0 //

ix. In a similar manner, create an extrusion of the other face of the torus segment to create the 200 mm inlet extension. You will probably find it helpful to rotate the view so that you can easily select the other face of the bend.

You can use the mouse buttons to change your view of the 3D image. The following table describes mouse actions that are available:

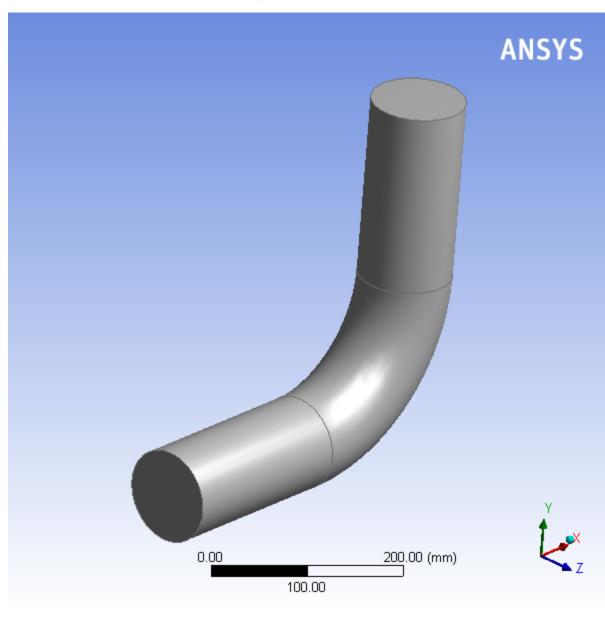
Table 1.1:	DesignModeler	View Manipu	lation Instructions
------------	---------------	-------------	---------------------

Action	Using Graphics Toolbar Buttons and the Mouse		
Rotate view (vertical, horizontal)	After clicking the <b>Rotate</b> icon, press and hold the left mouse button and drag the mouse. Dragging side to side rotates the view about the vertical axis, and dragging up and down rotates the view about the horizontal axis.		
Translate or pan view	After clicking the <b>Pan</b> icon, , press and hold the left mouse button and drag the object with the mouse until the view is satisfactory.		

Action	Using Graphics Toolbar Buttons and the Mouse		
Zoom in and out of view	After clicking the <b>Zoom</b> icon, (I), press and hold the left mouse button and drag the mouse up and down to zoom in and out of the view.		
Box zoom	After clicking the <b>Box Zoom</b> icon, $$ , press and hold the left mouse button and drag the mouse diagonally across the screen. This action will cause a rectangle to appear in the display. When you release the mouse button, a new view will be displayed that consists entirely of the contents of the rectangle.		

Clicking the **Zoom to Fit** icon, will cause the object to fit exactly and be centered in the window.

After entering the extrusion parameters and clicking **Generate**, the geometry should appear as in Figure 1.5: Elbow Main Pipe Geometry (p. 16).



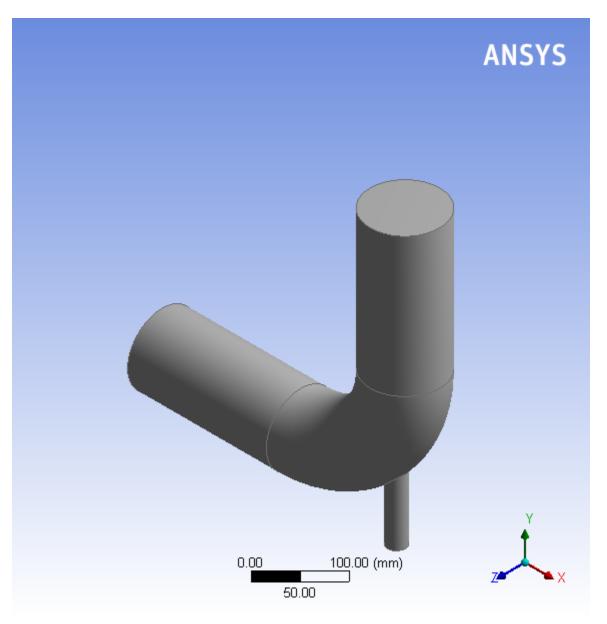
#### Figure 1.5: Elbow Main Pipe Geometry

- b. Next you will use a cylinder primitive to create the side pipe.
  - i. Choose **Create**  $\rightarrow$  **Primitives**  $\rightarrow$  **Cylinder** from the menubar.
  - ii. In the **Details View**, set the parameters for the cylinder as follows and click **Generate**:

Setting	Value
BasePlane	XYPlane
FD3, Origin X Coordinate	137.5
FD4, Origin Y Coordinate	-225
FD5, Origin Z Coordinate	0
FD6, Axis X Component	0
FD7, Axis Y Component	125

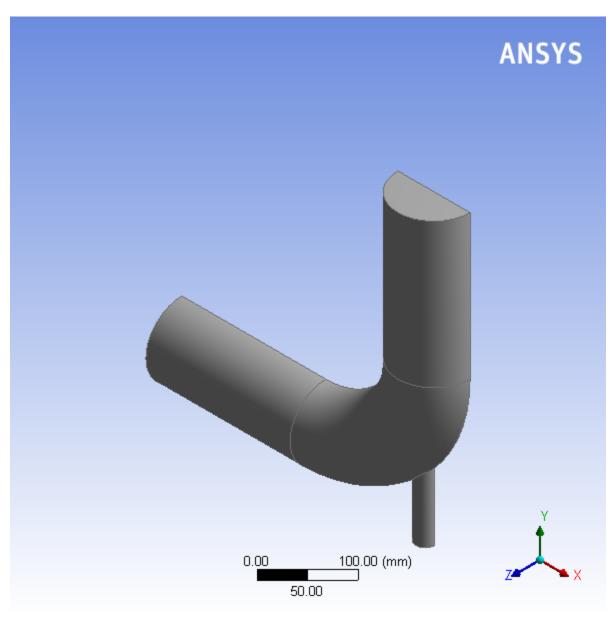
Setting	Value
FD8, Axis Z Component	0
FD10, Radius (>0)	12.5

The Origin Coordinates determine the starting point for the cylinder and the Axis Components determine the length and orientation of the cylinder body.



- c. The final step in creating the geometry is to split the body on its symmetry plane which will halve the computational domain.
  - i. Choose **Tools**  $\rightarrow$  **Symmetry** from the menu bar.
  - ii. Select the XYPlane in the Tree Outline.
  - iii. Click Apply next to Symmetry Plane 1 in the Details view.

iv. Click Generate.



The new surface created with this operation will be assigned a symmetry boundary condition in Fluent so that the model will accurately reflect the physics of the complete elbow geometry even though only half of it is meshed.

- d. Specify the geometry as a fluid body.
  - i. In the Tree Outline, open the 1 Part, 1 Body branch and select Solid.
  - ii. In the **Details View** of the body, change the name of the **Body** from **Solid** to **Fluid**.
  - iii. In the Fluid/Solid section, select Fluid from the drop-down list.

De	etails View	д
Ξ	Details of Body	
	Body	Fluid
	Volume	2.5159e+006 mm3
	Surface Area	1.7636e+005 mm <sup>2</sup>
	Faces	8
	Edges	18
	Vertices	12
	Fluid/Solid	Fluid
	Shared Topology Method	Automatic
	Geometry Type	DesignModeler

#### iv. Click Generate.

## Tip

In addition to the primitives you used in this tutorial, ANSYS DesignModeler offers a full suite of 2D sketching and 3D solid modeling tools for creating arbitrary geometry. Refer to the DesignModeler User's Guide for more information.

- 4. Close ANSYS DesignModeler by selecting File → Close DesignModeler or by clicking the 'X' icon in the upper right-hand corner. ANSYS Workbench automatically saves the geometry and updates the Project Schematic accordingly. The question mark in the Geometry cell is replaced by a check mark, indicating that there is a geometry now associated with the fluid flow analysis system.
- 5. View the list of files generated by ANSYS Workbench by selecting **View**  $\rightarrow$  **Files**.

Figure 1.6: ANSYS Workbench Files View for the Project After Creating the Geometry

Files	Files										
	А	В	с	D	E						
1	Name 💽 C		Size 💌	Туре 💌	Date Modified 💌	Location					
2	🔥 elbow-workbench.wbpj		124 KB	ANSYS Project File	XX/XX/20XX 0:00:00 AM	C:\Tutorial_01					
3	♥ FFF.agdb A2		2 MB	Geometry File	XX/XX/20XX 0:00:00 AM	C:\Tutorial_01\elbow-wor					
4	🂐 designPoint.wbdp		26 KB	Design Point File	XX/XX/20XX 0:00:00 AM	C:\Tutorial_01\elbow-woi					

Note the addition of the geometry file (FFF.agdb, where FFF indicates a Fluent-based fluid flow system) to the list of files. If you had imported the geometry file provided for this tutorial rather than creating the geometry yourself, the elbow\_geometry.agdb (or the elbow\_geometry.stp) file would be listed instead.

# 1.4.4. Meshing the Geometry in the ANSYS Meshing Application

Now that you have created the mixing elbow geometry, you must generate a computational mesh throughout the flow volume. For this section of the tutorial, you will use the ANSYS Meshing application to create a mesh for your CFD analysis, then review the list of files generated by ANSYS Workbench.

#### Important

Note the **Refresh Required** icon ( $\gtrless$ ) within the **Mesh** cell for the system. This indicates that the state of the cell requires a refresh and that upstream data has changed since the last refresh or update (such as an update to the geometry). Once the mesh is defined, the state of the **Mesh** cell will change accordingly, as will the state of the next cell in the system, in this case the **Setup** cell.

•		А	
1		Fluid Flow (Fluent)	
2	m	Geometry	× .
3	6	Mesh	2 🖌
4	٢	Setup	7
5	1	Solution	? 🖌
6	6	Results	7
		elbow	

1. Open the ANSYS Meshing application.

In the ANSYS Workbench **Project Schematic**, double-click the **Mesh** cell in the elbow fluid flow analysis system (cell A3). This displays the ANSYS Meshing application with the elbow geometry already loaded. You can also right-click the **Mesh** cell to display the context menu where you can select the **Edit...** option.

😭 A : elbow - Meshing [A	NSYS ICEM CFD]				
File Edit View Units	Tools Help	🥩 Generate Mesh  🏙 🔥	🙋 👻 🌒 Worksheet 👘	i n	
🚏 🦞 💽 • 🖏 • [	te te te te 🧣	) - 'S 🕂 Q 🕀 🔍 🔍	QQ 💥 🎊 🙆	) 🔒 🗞 🗖 🗸 👘	
		oloring + 10 + 11 + 12 + 13			nw Mesh 🛛 🙏
		Connections Tracture			
Outline	4		,		
Filter: Name -				A 1	ICVC
Project				Ar	ISYS
Geometry     Coordinate	Systems				
Details of "Model"	<del>.</del>				
Ambient 0.1					
Diffuse 0.6					
Specular 1 Color					
		0.000	0.100	1.200 (m)	Ý ×
	Geometry	0.000		).200 (m) ⊐ 2≝	Y X

Figure 1.7: The ANSYS Meshing Application with the Elbow Geometry Loaded

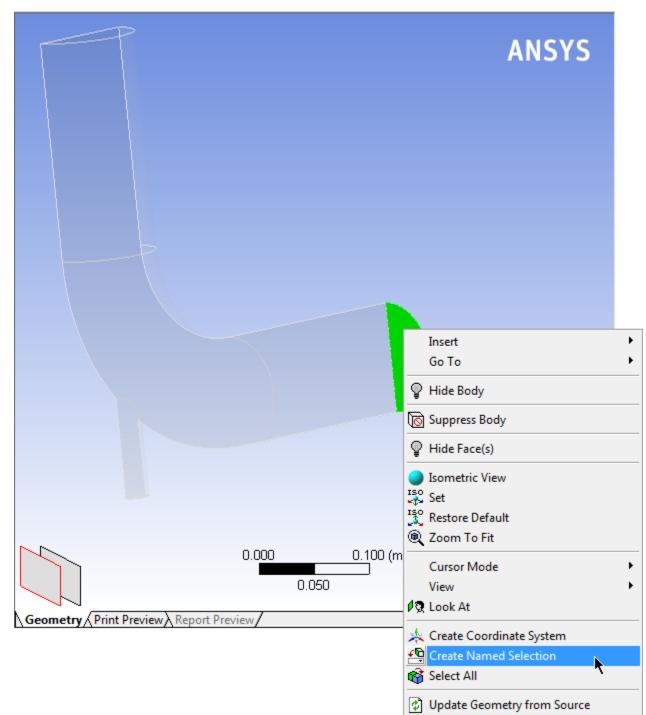
2. Create named selections for the geometry boundaries.

In order to simplify your work later on in ANSYS Fluent, you should label each boundary in the geometry by creating named selections for the pipe inlets, the outlet, and the symmetry surface (the outer wall boundaries are automatically detected by ANSYS Fluent).

a. Select the large inlet in the geometry that is displayed in the ANSYS Meshing application.

#### Tip

- Use the Graphics Toolbar buttons and the mouse to manipulate the image until you can easily see the pipe openings and surfaces.
- To select the inlet, the *Single select* ( ) mode must be active.
- b. Right-click and select the **Create Named Selection** option.



## Figure 1.8: Selecting a Face to Name

This displays the **Selection Name** dialog box.

Selection Name
velocity-inlet-large
Apply selected geometry
Apply geometry items of same:
Size
П Туре
Location X
Location Y
Location Z
Apply To Corresponding Mesh Nodes
OK Cancel

Figure 1.9: Applying a Name to a Selected Face

- c. In the **Selection Name** dialog box, enter velocity-inlet-large for the name and click **OK**.
- d. Perform the same operations for:
  - The small inlet (velocity-inlet-small)
  - The large outlet (pressure-outlet)
  - The symmetry plane (symmetry).

The named selections that you have created appear under the **Named Selections** item in the **Outline** view.

#### Important

It is important to note that by using the strings "velocity inlet" and "pressure outlet" in the named selections (with or without hyphens or underscore characters), ANSYS Fluent automatically detects and assigns the corresponding boundary types accordingly.

- 3. Create a named selection for the fluid body.
  - a. Change the selection filter to **Body** in the **Graphics Toolbar** (**D**).
  - b. Click the elbow in the graphics display to select it.
  - c. Right-click, select the Create Named Selection option and name the body Fluid.

By creating a named selection called Fluid for the fluid body you will ensure that ANSYS Fluent automatically detects that the volume is a fluid zone and treats it accordingly.

4. Set basic meshing parameters for the ANSYS Meshing application.

For this analysis, you will adjust several meshing parameters to obtain a finer mesh.

a. In the **Outline** view, select **Mesh** under **Project/Model** to display the **Details of "Mesh"** view below the **Outline** view.

#### Important

Note that because the ANSYS Meshing application automatically detects that you are going to perform a CFD fluid flow analysis using ANSYS Fluent, the **Physics Preference** is already set to **CFD** and the **Solver Preference** is already set to **Fluent**.

- Expand the Sizing node by clicking the "+" sign to the left of the word Sizing to reveal additional sizing parameters. Change Relevance Center to Fine by clicking the default value, Coarse, and selecting Fine from the drop-down list.
- c. Expand the **Quality** node to reveal additional quality parameters. Change **Smoothing** to **High**.
- d. Add a Body Sizing control.
  - i. With **Mesh** still selected in the **Outline** tree, click the elbow in the graphics display to select it.
  - ii. Right click in the graphics area and select **Insert**  $\rightarrow$  **Sizing** from the context menu.

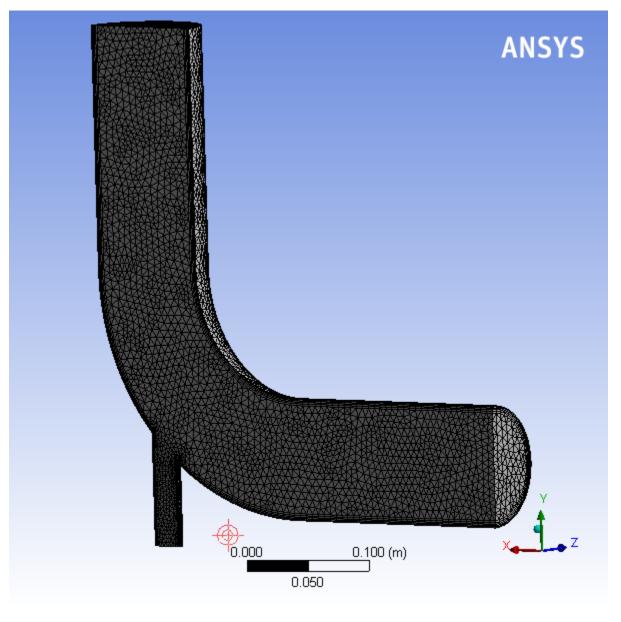
			ANSYS
	Insert		🕲 Method
	Go To	•	
	<ul> <li>Generate Mesh On Selected Bodies</li> <li>Preview Surface Mesh On Selected Bodies</li> <li>Clear Generated Data On Selected Bodies</li> </ul>		Image: Sizing       Image: Sizing       Image: Sizing       Image: Sizing
	Parts © Hide Body	•	-
	<ul> <li>Suppress Body</li> <li>Isometric View</li> <li>Set</li> <li>Restore Default</li> <li>Zoom To Fit</li> </ul>		
1	Cursor Mode View V Look At	* *	Y A
đ	k Create Coordinate System Create Named Selection		Xanta Z

A new Body Sizing entry appears under Mesh in the project Outline tree

- iii. In the **Outline** tree, click the new **Body Sizing** control.
- iv. Enter 6e-3 for Element Size and press Enter.
- e. Click again **Mesh** in the **Outline** view and in the **Details of "Mesh"** view, expand the **Inflation** node to reveal additional inflation parameters. Change **Use Automatic Inflation** to **Program Controlled**.
- 5. Generate the mesh.

Right-click **Mesh** in the project **Outline** tree, and select **Update** in the context menu.

#### Figure 1.10: The Computational Mesh for the Elbow Geometry in the ANSYS Meshing Application



#### Important

Using the **Generate Mesh** option creates the mesh, but does not actually create the relevant mesh files for the project and is optional if you already know that the mesh is acceptable. Using the **Update** option automatically generates the mesh, creates the relevant mesh files for your project, and updates the ANSYS Workbench cell that references this mesh.

#### Note

Once the mesh is generated, you can view the mesh statistics by opening the **Statistics** node in the **Details of "Mesh"** view. This will display information such as the number of nodes and the number of elements.

6. Close the ANSYS Meshing application.

You can close the ANSYS Meshing application without saving it because ANSYS Workbench automatically saves the mesh and updates the **Project Schematic** accordingly. The **Refresh Required** icon in the **Mesh** cell has been replaced by a check mark, indicating that there is a mesh now associated with the fluid flow analysis system.

7. View the list of files generated by ANSYS Workbench.

#### $View \rightarrow Files$

Figure 1.11: ANSYS Workbench Files View for the Project After Mesh Creation
---

Files	Files										
	А	В	С	D	E						
1	Name 💌	Cell 🔽 ID	Size 💌	Туре 💌	Date Modified 💌	Location					
2	🔥 elbow-workbench.wbpj		122 KB	ANSYS Project File	XX/XX/20XX 0:00:00 AM	C:\Tutorial_01					
3	🥪 FFF.agdb	A2	1 MB	Geometry File	XX/XX/20XX 0:00:00 AM	C:\Tutorial_01\elbow-wo					
4	FFF.msh	A3	8 MB	Fluent Mesh File	XX/XX/20XX 0:00:00 AM	C:\Tutorial_01\elbow-wo					
5	🗑 FFF.mshdb	A3	2 MB	Mesh Database Files	XX/XX/20XX 0:00:00 AM	C:\Tutorial_01\elbow-wo					
6	🌂 designPoint.wbdp		25 KB	Design Point File	XX/XX/20XX 0:00:00 AM	C:\Tutorial_01\elbow-wo					

Note the addition of the mesh files (FFF.msh and FFF.mshdb) to the list of files. The FFF.msh file is created when you update the mesh, and the FFF.mshdb file is generated when you close the ANSYS Meshing application.

# 1.4.5. Setting Up the CFD Simulation in ANSYS Fluent

Now that you have created a computational mesh for the elbow geometry, you will set up a CFD analysis using ANSYS Fluent, then review the list of files generated by ANSYS Workbench.

# 1.4.5.1. Starting ANSYS Fluent

1. In the ANSYS Workbench **Project Schematic**, double-click the **Setup** cell in the elbow fluid flow analysis system. You can also right-click the **Setup** cell to display the context menu where you can select the **Edit...** option.

When ANSYS Fluent is first started, the Fluent Launcher is displayed, enabling you to view and/or set certain ANSYS Fluent start-up options.

#### Note

The Fluent Launcher allows you to decide which version of ANSYS Fluent you will use, based on your geometry and on your processing capabilities.

## Figure 1.12: Fluent Launcher

Fluent Launcher (Setting Edit Only)	
<b>ANSYS</b>	Fluent Launcher
Dimension © 2D @ 3D	Options Double Precision  Meshing Mode
Display Options ☑ Display Mesh After Reading ☑ Workbench Color Scheme ▣ Do not show this panel again	Processing Options Serial Parallel
ACT Option  Load ACT	
💽 Show More Options	
	ncel <u>H</u> elp ▼

2. Ensure that the proper options are enabled.

#### Important

Note that the **Dimension** setting is already filled in and cannot be changed, since ANSYS Fluent automatically sets it based on the mesh or geometry for the current system.

- a. Ensure that **Serial** from the **Processing Options** list is enabled.
- b. Select Double Precision under Options.

c. Ensure that the Display Mesh After Reading and Workbench Color Scheme options are enabled.

#### Note

An option is enabled when there is a check mark in the check box, and disabled when the check box is empty. To change an option from disabled to enabled (or vice versa), click the check box or the text.

#### Note

Fluent will retain your preferences for future sessions.

3. Click OK to launch ANSYS Fluent.

#### Note

The ANSYS Fluent settings file (FFF.set) is written as soon as ANSYS Fluent opens.

Figure 1.13: The ANSYS Fluent Application

File 🙀 Setting Up Domain	A Setting Up Physics	User Defined	Solving 🦻 Postpr	rocessing V	Newing Parallel D	esion 🙆	2 DB _ AN
•	Scale alty Transform ove Make Polyhedra	Combine Delete. Separate Deactivat Adjacency Activate	Append	Interfaces Mesh Overset	Mesh Models Dynamic Nesh Moing Planes Turbo Topology	Adapt Mark/Adapt Cells Manape Registers Nore	Surface Create Manage
14	Task Page	×		Mesh	8		
Stup General Gener	General Mesh ScaleOne Display Solver Type © Pressure-Based © Densty-Based Time © Staady © Transient © Gravity <u>Units</u> Heb	ck Report Quality Veboty Formulation @ Absolute © Relative	S		Done -		À

# 1.4.5.2. Setting Up Domain

In this step, you will perform the mesh-related activities using the **Setting Up Domain** ribbon tab (**Mesh** group).

Introduction to Using ANSYS Fluent in ANSYS Workbench: Fluid Flow and Heat Transfer in a Mixing Elbow

	File		Setti	ng Up D	omain		Setting Up Physics
				P	1esh		
1	🗊 Display	y		1	*	8	Scale
	Info		-	Check	Qualit	a Y	Transform
	Units			Repair	Improve	e	Make Polyhedra

#### Note

These controls are also available in the **General** task page that can be accessed by clicking the **Setup/General** tree item.

1. Change the units for length.

Because you want to specify and view values based on a unit of length in millimeters from within ANSYS Fluent, change the units of length within ANSYS Fluent from meters (the default) to millimeters.

#### Important

Note that the ANSYS Meshing application automatically converts and exports meshes for ANSYS Fluent using meters (m) as the unit of length regardless of what units were used to create them. This is so you do not have to scale the mesh in ANSYS Fluent under ANSYS Workbench.

# **Setting Up Domain** $\rightarrow$ Mesh $\rightarrow$ Units...

This displays the **Set Units** dialog box.

Set Units				<b>—</b> ×
Quantities kinematic-viscosity length length-inverse length-time-inverse mag-permeability mass		^	Units m cm mm in ft	Set All to default si british
mass-diffusivity mass-flow mass-flow-per-depth mass-flux mass-transfer-rate mole-transfer-rate		Ŧ	Factor 0.001 Offset 0	
	New List		Close Help	

- a. Select length in the Quantities list.
- b. Select **mm** in the **Units** list.

c. Close the dialog box.

#### Note

Now, all subsequent inputs that require a value based on a unit of length can be specified in millimeters rather than meters.

2. Check the mesh.

```
Setting Up Domain 
ightarrow Mesh 
ightarrow Check
```

#### Note

ANSYS Fluent will report the results of the mesh check in the console.

```
Domain Extents:
    x-coordinate: min (m) = -2.000000e-01, max (m) = 2.000000e-01
    y-coordinate: min (m) = -2.250000e-01, max (m) = 2.000000e-01
    z-coordinate: min (m) = 0.000000e+00, max (m) = 5.000000e-02
Volume statistics:
    minimum volume (m3): 1.144763e-10
    maximum volume (m3): 5.871098e-08
    total volume (m3): 2.511309e-03
Face area statistics:
    minimum face area (m2): 2.051494e-07
    maximum face area (m2): 3.429518e-05
Checking mesh......
Done.
```

#### Note

The minimum and maximum values may vary slightly when running on different platforms. The mesh check will list the minimum and maximum x and y values from the mesh in the default SI unit of meters. It will also report a number of other mesh features that are checked. Any errors in the mesh will be reported at this time. Ensure that the minimum volume is not negative as ANSYS Fluent cannot begin a calculation when this is the case.

3. Review the mesh quality.

```
Setting Up Domain \rightarrow Mesh \rightarrow Quality
```

#### Note

ANSYS Fluent will report the results of the mesh quality below the results of the mesh check in the console.

```
Mesh Quality:
Minimum Orthogonal Quality = 1.98483e-01
(Orthogonal Quality ranges from 0 to 1, where values close to 0 correspond to low quality.)
Maximum Ortho Skew = 7.19694e-01
```

Introduction to Using ANSYS Fluent in ANSYS Workbench: Fluid Flow and Heat Transfer in a Mixing Elbow

(Ortho Skew ranges from 0 to 1, where values close to 1 correspond to low quality.) Maximum Aspect Ratio = 2.05643e+01

#### Note

The quality of the mesh plays a significant role in the accuracy and stability of the numerical computation. Checking the quality of your mesh is, therefore, an important step in performing a robust simulation. Minimum cell orthogonality is an important indicator of mesh quality. Values for orthogonality can vary between 0 and 1 with lower values indicating poorer quality cells. In general, the minimum orthogonality should not be below 0.01 with the average value significantly larger. The high aspect ratio cells in this mesh are near the walls and are a result of the boundary layer inflation applied in the meshing step. For more information about the importance of mesh quality refer to the Fluent User's Guide.

# 1.4.5.3. Setting Up Physics

10001 10001 10

In the steps that follow, you will select a solver and specify physical models, material properties, and zone conditions for your simulation using the **Setting Up Physics** ribbon tab.

1. In the **Solver** group of the **Setting Up Physics** ribbon tab, retain the default selection of the steady pressurebased solver.

S	etting Up Physics $ ightarrow$ S	olver			
File	🔹 Setting Up Domain	🤹 Setting Up	Physics	User Defined - 🍯	
		Solver			
Time Type Steady Pressure-Based		Velocity Formulation d	Operati	Operating Conditions	
Trar	nsient 🔘 Density-Based	l 🔘 Relative	Reference Values		

2. Set up your models for the CFD simulation using the **Models** group of the **Setting Up Physics** ribbon tab.

		Models	
	Radiation	💽 Multiphase	🙆 Solidify/Melt
Energy	∦ <sub>‡</sub> Heat Exchanger	Species	(1)) Acoustics
	So Viscous	Siscrete Phase	🗄 More 🗸

#### Note

You can also use the **Models** task page that can be accessed by double-clicking the **Setup/Models** tree item.

a. Enable heat transfer by activating the energy equation.

In the Setting Up Physics ribbon tab, select Energy (Models group).

# 

- b. Enable the k- $\varepsilon$  turbulence model.
  - i. In the Setting Up Physics ribbon tab, click Viscous... (Models group).

Setting Up Physics  $\rightarrow$  Models  $\rightarrow$  Viscous...

Viscous Model	<b>—</b>
Model	Model Constants
Inviscid	Cmu 📩
🔘 Laminar	0.09
Spalart-Allmaras (1 eqn)	C1-Epsilon
k-epsilon (2 eqn)	1.44
k-omega (2 eqn)	C2-Epsilon
<ul> <li>Transition k-kl-omega (3 eqn)</li> <li>Transition SST (4 eqn)</li> </ul>	1.92
<ul> <li>Reynolds Stress (7 eqn)</li> </ul>	TKE Prandtl Number
<ul> <li>Scale-Adaptive Simulation (SAS)</li> </ul>	1
Detached Eddy Simulation (DES)	TOD Description
$\odot$ Large Eddy Simulation (LES)	User-Defined Functions
-k-epsilon Model-	Turbulent Viscosity
<ul> <li>Standard</li> </ul>	none 🔻
© RNG	Prandtl Numbers
Realizable	TKE Prandtl Number
Near-Wall Treatment	none 🔻
Standard Wall Functions	TDR Prandtl Number
Scalable Wall Functions	none 🔻
Non-Equilibrium Wall Functions	Energy Prandtl Number
Enhanced Wall Treatment	none
Menter-Lechner	Wall Prandtl Number
O User-Defined Wall Functions	none 👻
Enhanced Wall Treatment Options	
Pressure Gradient Effects	
Thermal Effects	
Options	
Viscous Heating	
Curvature Correction	
Production Kato-Launder	
Production Limiter	
ОК	Cancel Help

ii. In the Viscous Model dialog box, select k-epsilon from the Model list.

#### Note

The **Viscous Model** dialog box will expand.

- iii. Use the default **Standard** from the **k-epsilon Model** group.
- iv. Select Enhanced Wall Treatment for the Near-Wall Treatment.

#### Note

The default Standard Wall Functions are generally applicable if the first cell center adjacent to the wall has a y+ larger than 30. In contrast, the Enhanced Wall Treatment option provides consistent solutions for all y+ values. Enhanced Wall Treatment is recommended when using the k-epsilon model for general single-phase fluid flow problems. For more information about Near Wall Treatments in the k-epsilon model refer to the Fluent User's Guide.

v. Click **OK** to accept the model and close the **Viscous Model** dialog box.

Note that the **Viscous...** label in the ribbon is displayed in blue to indicate that the Viscous model is enabled.

3. Set up the materials for your CFD simulation using the **Materials** group of the **Setting Up Physics** ribbon tab.



Create a new material called **water** using the **Create/Edit Materials** dialog box (Figure 1.14: The Create/Edit Materials Dialog Box (p. 35)).

a. In the Setting Up Physics ribbon tab, click Create/Edit... (Materials group).

Setting Up Physics  $\rightarrow$  Materials  $\rightarrow$  Create/Edit...

- b. In the Create/Edit Materials dialog box, type water for Name.
- c. In the **Properties** group, enter the following values:

Property	Value
Density	1000 [kg/m <sup>3</sup> ]
<i>c<sub>p</sub></i> (Specific Heat)	4216 [J/kg-K]
Thermal Conductivity	0.677 [W/m-K]

Property	Value
Viscosity	8e-04 [ <b>kg/m-s</b> ]

lame	Mahadal Tura		Order Materials by
water	Material Type fluid		
hemical Formula	Fluent Fluid Materials		Chemical Formula
	water		Fluent Database
	Mixture		User-Defined Database
	none		-
roperties			_
Density (kg/m3) const	ant 🗸 Edit	n é	
Const		211	
1000			
Cp (Specific Heat) (j/kg-k) const	ant 👻 Edit		
4216			
Thermal Conductivity (w/m-k) const	ant 👻 Edit		
0.67			
Viscosity (kg/m-s) const	ant 💌 Edit		
Const		21	
0.00	38	-	

d. Click Change/Create.

A **Question** dialog box will open, asking if you want to overwrite air. Click **No** so that the new material **water** is added to the **Fluent Fluid Materials** list of materials that originally contained only **air**.

Question		×
?	Change/Create mixture and Overwrite air?	
	Yes No	

#### Extra

You could have copied the material **water-liquid** (h2o < l >) from the materials database (accessed by clicking the **ANSYS Fluent Database...** button). If the properties in the database are different from those you want to use, you can edit the values in the **Properties** group in the **Create/Edit Materials** dialog box and click **Change/Create** to update your local copy. The original copy will not be affected. Introduction to Using ANSYS Fluent in ANSYS Workbench: Fluid Flow and Heat Transfer in a Mixing Elbow

e. Ensure that there are now two materials (water and air) defined locally by examining the **Fluent Fluid Materials** drop-down list in the **Create/Edit Materials** dialog box .

#### Note

Both the materials will also be listed under **Fluid** in the **Materials** task page and under the **Materials** tree branch.

- f. Close the Create/Edit Materials dialog box.
- 4. Set up the cell zone conditions for the CFD simulation using the **Zones** group of the **Setting Up Physics** ribbon tab.

Zones
Cell Zones
Boundaries
Profiles

a. In the Setting Up Physics tab, click Cell Zones.

**Setting Up Physics**  $\rightarrow$  **Zones**  $\rightarrow$  **Cell Zones** 

This opens the Cell Zone Conditions task page.

Cell Z	one C	ondit	ions					
Zone	Filter	Text					-0	-
fluid								
Par Disp Porou	Edit ameter lay Me us Form uperfic	sh) nulatio	Co Ope n	py eratin <u>c</u>	Profi Cond	ID -1 les	]	
	hysical							

- b. Set the cell zone conditions for the fluid zone.
  - i. In the **Cell Zone Conditions** task page, in the **Zone** list, select **fluid** and click **Edit...** to open the **Fluid** dialog box.

## Note

You can also double-click **fluid** in the **Cell Zones Conditions** task page or under the **Setup/Cell Zone Conditions** tree branch in order to open the corresponding dialog box.

Introduction to Using ANSYS Fluent in ANSYS Workbench: Fluid Flow and Heat Transfer in a Mixing Elbow

Porous Zone         Reference Frame       Mesh Motion       Porous Zone       3D Fan Zone       Embedded LES       Reaction         Rotation-Axis Origin       Rotation-Axis Direction <ul> <li>X (mm)</li> <li>0</li> <li>constant</li> <li>7 (mm)</li> <li>0</li> <li>constant</li> <li>7 (mm)</li> <li>0</li> <li>constant</li> <li>7 (mm)</li> <li>constant</li> <li>2 (mm)</li> <li>0</li> <li>constant</li> <li>7 (mm)</li> <li>constant</li> <li>2 (mm)</li> <li>constant</li> <li>model</li> <li>model</li></ul>	
X (mm)         0         constant         -         X         0         constant           Y (mm)         0         constant         -         Y         0         constant	ource Terms   Fixed Values   Multipha
Y (mm) 0 constant V 0 constant	*
	•
Z (mm) 0 constant	•
	•

- ii. In the Fluid dialog box, select water from the Material Name drop-down list.
- iii. Click **OK** to close the **Fluid** dialog box.
- 5. Set up the boundary conditions for the CFD analysis using the **Zones** group of the **Setting Up Physics** ribbon tab.

Zones
Cell Zones
Boundaries
Profiles

a. In the Setting Up Physics tab, click Boundaries (Zones group).

**Setting Up Physics**  $\rightarrow$  **Zones**  $\rightarrow$  **Boundaries** 

This opens the **Boundary Conditions** task page.

Boundary Conditions
Zone Filter Text
<ul> <li>Inlet         velocity-inlet-large         velocity-inlet-small</li> <li>Internal         interior-fluid</li> <li>Outlet         pressure-outlet</li> <li>Symmetry         symmetry         symmetry</li> <li>Wall         wall-fluid</li> </ul>
Phase Type ID mixture > -1
Edit       Copy       Profiles         Parameters       Operating Conditions         Display Mesh       Periodic Conditions
Highlight Zone
Help

#### Note

To display boundary zones grouped by zone type (as shown above), click the **Toggle Tree View** button in the upper right corner of the **Boundary Conditions** task page and under the **Group By** category select **Zone Type** from the drop-down list.

b. Set the boundary conditions at the cold inlet (velocity-inlet-large).

## Tip

If you are unsure of which inlet zone corresponds to the cold inlet, you can use the mouse to probe for mesh information in the graphics window. If you click the right mouse button with the pointer on any node in the mesh, information about the associated zone will be displayed in the ANSYS Fluent console, including the name of the zone. The zone you probed will be automatically selected from the **Zone** selection list in the **Boundary Conditions** task page.

Alternatively, you can click the probe button ( ) in the graphics toolbar and click the left mouse button on any node. This feature is especially useful when you have several zones of the same type and you want to distinguish between them quickly. The information will be displayed in the console.

i. From the Zone selection list, select velocity-inlet-large and click Edit....

Velocity Inlet										
Zone Name										
velocity-inlet-large										
Momentum Thermal Radiation Species DPM Multiphase Potential UDS										
Velocit	y Specification	n Method Ma	gnitude, No	rmal to B	oundary		•			
	Referen	ce Frame Ab	solute				-			
Velocity Magnitude (m/s) 0.4 constant										
Supersonic/Initial Gauge Pressure (pascal) 0 constant										
Turbulence										
Specification Method Intensity and Hydraulic Diameter										
Turbulent Intensity (%) 5										
Hydraulic Diameter (mm) 100										
OK Cancel Help										

- ii. In the Velocity Inlet dialog box, ensure that Magnitude, Normal to Boundary is selected for Velocity Specification Method.
- iii. Enter 0.4 m/s for **Velocity Magnitude**.
- iv. In the **Turbulence** group, from the **Specification Method** drop-down list, select **Intensity and Hydraulic Diameter**.
- v. Retain the default of 5 % for Turbulent Intensity.
- vi. Enter 100 mm for Hydraulic Diameter.

#### Note

The hydraulic diameter  $D_h$  is defined as:

$$D_h = \frac{4A}{P_w}$$

where A is the cross-sectional area and  $P_w$  is the wetted perimeter.

## vii. Click the **Thermal** tab.

Velocity Inlet										
Zone Name										
velocity-inlet-large										
Momentum	Thermal	Radiation	Species	DPM	Multiphase	Potential	UDS			
Temperature (	k) 293.15		consta	nt	•					
OK Cancel Help										

viii.Enter 293.15 *K* for **Temperature**.

- ix. Click **OK** to close the **Velocity Inlet** dialog box.
- c. In a similar manner, set the boundary conditions at the hot inlet (**velocity-inlet-small**), using the values in the following table:

Setting	Value
Velocity Specification Method	Magnitude, Normal to Boundary
Velocity Magnitude	1.2 [m/s]
Specification Method	Intensity & Hydraulic Diameter
Turbulent Intensity	5%
Hydraulic Diameter	25 [mm]
Temperature	313.15[K]

d. Double-click **pressure-outlet** and set the boundary conditions at the outlet, as shown in the **Pressure Outlet** dialog box. Introduction to Using ANSYS Fluent in ANSYS Workbench: Fluid Flow and Heat Transfer in a Mixing Elbow

Pressure Outlet										
Zone Name	Zone Name									
pressure-outlet	pressure-outlet									
Momentum Thermal Radiation Species DPM Multiphase Potential UDS										
Backf	low Refer	ence Frame	Absolute							
	Gauge Pr	ressure (pas	cal) 0		СО	nstant	•			
Backflow Direction	Specificat	ion Method	Normal to B	oundary			•			
Backflow Pressure Specification Total Pressure										
Radial Equilibrium Pressure Distribution										
Average Pressure Specification										
Target Mass Flow Rate										
Turbulence										
Specification Method Intensity and Hydraulic Diameter										
	Backflow Turbulent Intensity (%) 5									
	Backflow Hydraulic Diameter (mm) 100									
	OK Cancel Help									

## Note

ANSYS Fluent will use the backflow conditions only if the fluid is flowing into the computational domain through the outlet. Since backflow might occur at some point during the solution procedure, you should set reasonable backflow conditions to prevent convergence from being adversely affected.

# 1.4.6. Solving

In the steps that follow, you will set up and run the calculation using the **Solving** ribbon tab.

#### Note

You can also use the task pages listed under the **Solution** branch in the tree to perform solution-related activities.

- 1. Set up solution parameters for the CFD simulation.
  - a. Change the Gradient method.
    - i. In the **Solving** ribbon tab, click **Methods...** (Solution group).





This will open the **Solution Methods** task page.

Solution Methods	
Pressure-Velocity Coupling	
Scheme	
SIMPLE	•
Spatial Discretization	
Gradient	*
Green-Gauss Node Based 🔹	
Pressure	
Second Order	
Momentum	Ξ
Second Order Upwind	
Turbulent Kinetic Energy	
First Order Upwind 🔹	
Turbulent Dissipation Rate	
First Order Upwind 🔹	
Transient Formulation	Ŷ
Non-Iterative Time Advancement	
Frozen Flux Formulation	
Pseudo Transient	
Warped-Face Gradient Correction	
High Order Term Relaxation Options	
Default	
Help	

- ii. In the **Spatial Discretization** group of the **Solution Methods** task page, change the **Gradient** to **Green-Gauss Node Based**. This gradient method is suggested for tetrahedral meshes.
- b. Enable the plotting of residuals during the calculation.
  - i. In the **Solving** ribbon tab, click **Residuals...** (**Reports** group).

**Solving**  $\rightarrow$  Reports  $\rightarrow$  Residuals...

Introduction to Using ANSYS Fluent in ANSYS Workbench: Fluid Flow and Heat Transfer in a Mixing Elbow

Repor	ts					
🖄 Residuals	Convergence					
Definitions	File Plot					
Residual Monitors						×
Options		Equations Residual	Monitor Ci	neck Convergence	Absolute Criteria	
Print to Console		continuity			0.001	Ĵ
Window 1 Curv				$\checkmark$	0.001	н
Iterations to Plot		y-velocity		<b>V</b>	0.001	
1000		z-velocity		<b>V</b>	0.001	-
		Residual Values			Convergence Cri	terion
Iterations to Store		Normalize		Iterations	absolute	•
		Compute Loca	l Scale			
	OK Plot	Renormaliz	e Ca	ncel Hel	lp 🛛	

In the **Residual Monitors** dialog box, ensure that **Plot** is enabled in the **Options** group.

- ii. Keep the default values for the **Absolute Criteria** of the **Residuals**, as shown in the **Residual Monitors** dialog box.
- iii. Click **OK** to close the **Residual Monitors** dialog box.

#### Note

By default, all variables will be monitored and checked by ANSYS Fluent as a means to determine the convergence of the solution.

c. Create a surface report definition at the outlet (pressure-outlet).

Solving  $\rightarrow$  Reports  $\rightarrow$  Definitions  $\rightarrow$  New  $\rightarrow$  Surface Report  $\rightarrow$  Facet Maximum

temp-outlet-0    Options Custom Vectors   Per Surface Vectors of   Average Over Custom Vectors   1 Custom Vectors   Field Variable   Report Files [0/0]   Static Temperature   Static Temperature   Surfaces Fiter Text   interior-fluid   pressure-outlet   symmetry   velocity-inlet-large   velocity-inlet-small   wall-fluid   Presurer Viot   Prequency 3   Print to Console   Highlight Surfaces   New Surface	Name		Report Type	
Per Surface   Average Over   1   1   Report Files [0/0]   Image: Surface Sector Sec	temp-outlet-0		Facet Maximum	
Per Surface Average Over Custom Vectors Field Variable  Field Variable  Static Temperature Surfaces Filter Text interior-fluid pressure-outlet symmetry velocity-inlet-large velocity-inlet-large velocity-inlet-small wall-fluid  Prequency 3 Print to Console Highlight Surfaces	Options			
1   Field Variable   Temperature   Static Temperature   Surfaces Fitter Text   interior-fluid   pressure-outlet   symmetry   velocity-inlet-large   velocity-inlet-small   wall-fluid	Per Surface			
Field Variable   Report Files [0/0]   Static Temperature   Static Temperature   Surfaces Fiter Text   interior-fluid   pressure-outlet   symmetry   velocity-inlet-large   velocity-inlet-small   wall-fluid			Custom Vectors	
Report Plots [0/0]     Static Temperature     Surfaces Filter Text     Surfaces Filter Text     interior-fluid        pressure-outlet   symmetry   velocity-inlet-large   velocity-inlet-small   wall-fluid     Create   Report File   Report Plot   Frequency 3   Print to Console     Highlight Surfaces	1		Field Variable	
Static Temperature Surfaces Filter Text Surfaces Filter Text Interior-fluid Pressure-outlet symmetry velocity-inlet-large velocity-inlet-small wall-fluid Create Report File Report File Frequency 3 Highlight Surfaces Highlight Surfaces	Report Files [0/0]		Temperature	
Report Plots [0/0]     Interior-fluid   pressure-outlet   symmetry   velocity-inlet-large   velocity-inlet-small   wall-fluid     Create   Image: Report File   Image: Report Plot   Frequency 3   Image: Print to Console     Image: Highlight Surfaces				
Report Plots [0/0]     Image: Create   Image: Report File   Image: Report Plot   Frequency 3   Image: Print to Console     Image: Print to Console <td></td> <td></td> <td>Surfaces Filter Text</td> <td>-,</td>			Surfaces Filter Text	-,
Report Plots [0/0]     symmetry   velocity-inlet-large   velocity-inlet-small   wall-fluid     Create   Report File   Report Plot   Frequency 3   Print to Console     Highlight Surfaces			interior-fluid	
velocity-inlet-large velocity-inlet-small wall-fluid	Papart Plats [0/0]		pressure-outlet	
Create Report File Report Plot Frequency 3 Print to Console Highlight Surfaces	report Plots [0/0]			
Create       Image: Report File       Image: Report Plot       Frequency 3       Image: Print to Console       Image: Highlight Surfaces				
Image: Construction of the second				
Image: Console       Image: Console         Image: Console       Image: Console				
Report Plot       Frequency 3       Print to Console       Highlight Surfaces	Create			
Report Plot       Frequency 3       Print to Console       Highlight Surfaces	Report File			
Frequency 3 C Highlight Surfaces				
Print to Console Highlight Surfaces				
			Highlight Surfaces	
	Create Output Paramete	IL		

- i. In the **Surface Report Definition** dialog box, enter temp-outlet-0 for the **Name**.
- ii. Under the Create group, enable Report File and Report Plot.

During a solution run, ANSYS Fluent will write solution convergence data in a report file and plot the solution convergence history in a graphics window.

It is good practice to monitor physical solution quantities in addition to equation residuals when assessing convergence.

iii. Set **Frequency** to 3 by clicking the up-arrow button.

This setting instructs ANSYS Fluent to update the plot of the surface report and write data to a file after every 3 iterations during the solution.

- iv. Select Temperature... and Static Temperature from the Field Variable drop-down lists.
- v. Select pressure-outlet from the Surfaces selection list.
- vi. Click OK to save the surface report definition and close the Surface Report Definition dialog box.

The new surface report definition temp-outlet-0 appears under the **Solution/Report Definitions** tree item. ANSYS Fluent also automatically creates the following items:

- temp-outlet-0-rfile (under the Solution/Monitors/Report Files tree branch)
- temp-outlet-0-rplot (under the Solution/Monitors/Report Plots tree branch)
- d. In the tree, double-click the **temp-outlet-0-rfile** (under the **Solution/Monitors/Report Files**) and examine the report file settings in the **Edit Report File** dialog box.

Name			
temp-outlet-0-rfile			
Available Report Definitions [0/0]	x	Selected Report Definitions [0/1]	
		temp-outlet-0	
	r	Add>>	
	L	< <remove< td=""><td></td></remove<>	
	_		
Output File Base Name		New - Edit	
Output File Base Name .\\temp-outlet-0-rfile.out	Browse	New  Edit	
	Browse	New  Edit	
.\\temp-outlet-0-rfile.out	Browse	New   Edit	
.\\temp-outlet-0-rfile.out Full File Name	Browse	New  Edit	
.\\temp-outlet-0-rfile.out Full File Name Get Data Every 3 🔃 Iteration		New  Edit	

The dialog box is automatically populated with data from the temp-outlet-0 report definition. The report that will be written in a report file during a solution is listed under **Selected Report Definitions**.

Retain the default output file name and click **OK**.

e. In the tree, double-click **temp-outlet-0-rplot** (under **Solution/Monitors/Report Plots**) and examine the report plot settings in the **Edit Report Plot** dialog box.

Edit Report Plot	×
Name	
temp-outlet-0-rplot	
Available Report Definitions [0/0]	Selected Report Definitions [0/1]
	temp-outlet-0
	Add>> < <remove< td=""></remove<>
Options Plot Window 2 Curves Axes	New  Edit
Get Data Every	
3 🗘 iteration 💌	
Plot Title temp-outlet-0-rplot	
X-Axis Label iteration	
Y-Axis Labelacet Maximum of temperature	
Print to Console	
OH	K Cancel Help

The dialog box is automatically populated with data from the temp-outlet-0 report definition. As the solution progresses, the report that is listed under **Selected Report Definitions** will be plotted in a graphics tab window with the title specified in **Plot Title**.

Retain the default names for Plot Title and Y-Axis Label and click OK.

#### Note

You can create report definitions for different boundaries and plot them in the same graphics window. However, report definitions in the same **Report Plot** must have the same units.

f. Initialize the flow field using the **Initialization** group of the **Solving** ribbon tab.

Solvin	Solving → Initialization								
	Initializ	zation							
Method		Patch							
O Hybrid	More Settings	Reset Statistics							
Standard	Options		t = 0						
		Reset DPM	Initialize						

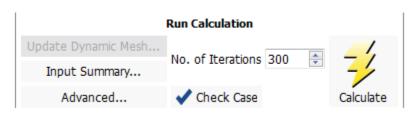
- i. Keep the **Method** at the default of **Hybrid**.
- ii. Click Initialize.
- g. Check to see if the case conforms to best practices.

## **Solving** $\rightarrow$ Run Calculation $\rightarrow$ Check Case

Case Check	×
Mesh Models Boundaries and Cell Zones Materials Solver	
Automatic Implementation	
Apply Recommendation Consider realizable k-epsilon in lieu of the standard k-epsilon turbulence model. (Models: Edit Viscous)	
Apply Close Help	

- i. Click the **Models** and **Solver** tabs and examine the **Recommendation** in each. These recommendations can be ignored for this tutorial. The issues they raise will be addressed in later tutorials.
- ii. Close the **Case Check** dialog box.
- 2. Calculate a solution using the **Run Calculation** group of the **Solving** tab.

## Solving → Run Calculation



- a. Start the calculation by requesting 300 iterations.
  - i. Enter 300 for **No. of Iterations**.
  - ii. Click Calculate.

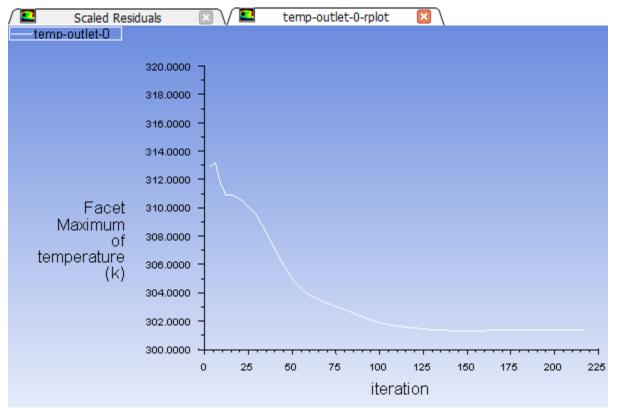
Note that while the program is calculating the solution, the states of the **Setup** and **Solution** cells in the fluid flow ANSYS Fluent analysis system in ANSYS Workbench are changing. For example:

• The state of the **Setup** cell becomes **Up-to-Date** and the state of the **Solution** cell becomes **Refresh Required** after the **Run Calculation** task page is visited and the number of iterations is specified.

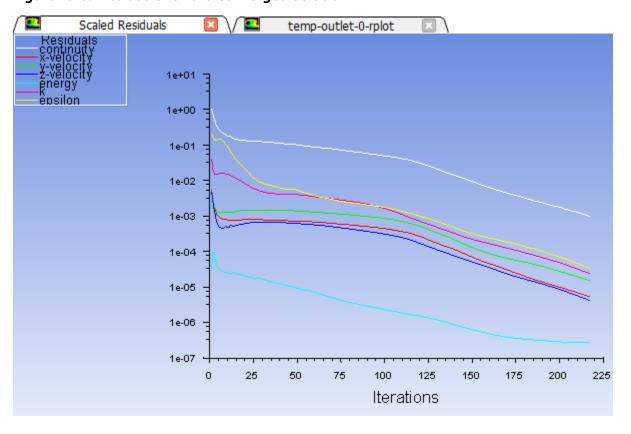
- The state of the **Solution** cell is **Update Required** while iterations are taking place.
- The state of the **Solution** cell is **Up-to-Date** when the specified number of iterations are complete (or if convergence is reached).

As the calculation progresses, the surface report plot will be plotted in the graphics window (**temp-outlet-0–rplot** tab window) (Figure 1.15: Convergence History of the Maximum Temperature at Pressure Outlet (p. 49)).





The residuals history will be plotted in the **Scaled Residuals** tab window (Figure 1.16: Residuals for the Converged Solution (p. 50)).



#### Figure 1.16: Residuals for the Converged Solution

#### Note

The solution will be stopped by ANSYS Fluent when the residuals reach their specified values or after 300 iterations. The exact number of iterations will vary depending on the platform being used. An **Information** dialog box will open to alert you that the calculation is complete. Click **OK** in the **Information** dialog box to proceed.

Because the residual values vary slightly by platform, the plot that appears on your screen may not be exactly the same as the one shown here.

Note that the ANSYS Fluent settings file (FFF.set) is updated in the **Files** view of the ANSYS Workbench before the calculation begins.

b. Examine the plots for convergence (Figure 1.16: Residuals for the Converged Solution (p. 50) and Figure 1.15: Convergence History of the Maximum Temperature at Pressure Outlet (p. 49)).

#### Note

There are no universal metrics for judging convergence. Residual definitions that are useful for one class of problem are sometimes misleading for other classes of problems. Therefore it is a good idea to judge convergence not only by examining residual levels, but also by monitoring relevant integrated quantities and checking for mass and energy balances.

There are three indicators that convergence has been reached:

• The residuals have decreased to a sufficient degree.

The solution has converged when the **Convergence Criterion** for each variable has been reached. The default criterion is that each residual will be reduced to a value of less than  $10^{-3}$ , except the **energy** residual, for which the default criterion is  $10^{-6}$ .

• The solution no longer changes with more iterations.

Sometimes the residuals may not fall below the convergence criterion set in the case setup. However, monitoring the representative flow variables through iterations may show that the residuals have stagnated and do not change with further iterations. This could also be considered as convergence.

• The overall mass, momentum, energy, and scalar balances are obtained.

You can examine the overall mass, momentum, energy and scalar balances in the **Flux Reports** dialog box. The net imbalance should be less than 0.2 % of the net flux through the domain when the solution has converged. In the next step you will check to see if the mass balance indicates convergence.

3. With ANSYS Fluent still running, go back to ANSYS Workbench and view the list of generated files.

#### View $\rightarrow$ Files

Note the addition of the surface report definition file temp-outlet-0.out to the list of files.

The status of the **Solution** cell is now up-to-date.

▼		А		
1		Fluid Flow (Fluent)		
2	œ	Geometry	× 🖌	
3	6	Mesh	× .	
4	٢	Setup	✓ ₄	
5	6	Solution	✓ ₄	
6	6	Results	2	
elbow				

# **1.4.7. Displaying Results in ANSYS Fluent and CFD-Post**

In this step, you will display the results of the simulation in ANSYS Fluent, display the results in CFD-Post, and then review the list of files generated by ANSYS Workbench.

1. Display results in ANSYS Fluent using the **Postprocessing** ribbon tab.

In ANSYS Fluent, you can perform a simple evaluation of the velocity and temperature contours on the symmetry plane. Later, you will use CFD-Post (from within ANSYS Workbench) to perform the same evaluation.

Introduction to Using ANSYS Fluent in ANSYS Workbench: Fluid Flow and Heat Transfer in a Mixing Elbow

a. Display filled contours of velocity magnitude on the symmetry plane (Figure 1.17: Velocity Distribution Along Symmetry Plane (p. 53)).

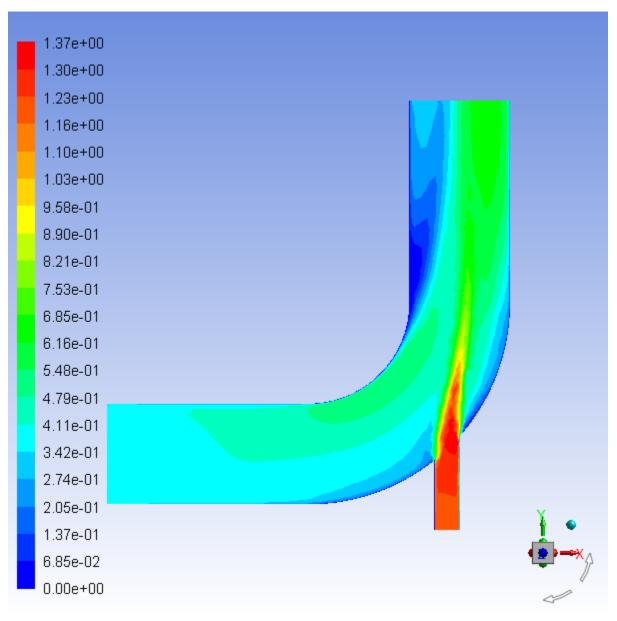
	Postprocessing $\rightarrow$	Graphics $\rightarrow$	Contours $\rightarrow$	Edit
--	------------------------------	------------------------	------------------------	------

#### Note

You can also double-click the Results/Graphics/Contours tree item.

Contours	
Options	Contours of
Filled	Velocity
Node Values Global Range	Velocity Magnitude
<ul> <li>Global Range</li> <li>Auto Range</li> </ul>	Min (m/s) Max (m/s)
Clip to Range	0 1.369013
Draw Profiles Draw Mesh	Surfaces Filter Text
	interior-fluid
Coloring	pressure-outlet
<ul> <li>Banded</li> </ul>	symmetry
Smooth	velocity-inlet-large
Smooth	velocity-inlet-small
Levels Setup	wall-fluid
20 💠 1 🚔	
	New Surface 🔻
	Display Compute Close Help

- i. In the Contours dialog box, in the Options group, enable Filled.
- ii. Ensure that **Node Values** is enabled in the **Options** group.
- iii. From the Contours of drop-down lists, select Velocity... and Velocity Magnitude.
- iv. From the **Surfaces** selection list, deselect all items by clicking 🖼 and then select **symmetry**.
- v. Click **Display** to display the contours in the active graphics window.





#### Note

You may want to clear Lighting in the Viewing ribbon tab (Display group).

b. Display filled contours of temperature on the symmetry plane (Figure 1.18: Temperature Distribution Along Symmetry Plane (p. 55)).

Postprocessing  $\rightarrow$  Graphics  $\rightarrow$  Contours  $\rightarrow$  Edit...

Introduction to Using ANSYS Fluent in ANSYS Workbench: Fluid Flow and Heat Transfer in a Mixing Elbow

Contours	
Options Filled	Contours of Temperature
Node Values Global Range	Static Temperature
Auto Range Clip to Range	Min Max 0 1.369013
Draw Profiles	Surfaces Filter Text
Calariaa	interior-fluid pressure-outlet
Coloring Banded Smooth Levels Setup	symmetry velocity-inlet-large velocity-inlet-small wall-fluid
20 🖈 1 📥	New Surface 🔻
	Display Compute Close Help

- i. Select **Temperature...** and **Static Temperature** from the **Contours of** drop-down lists.
- ii. Click **Display** and close the **Contours** dialog box.

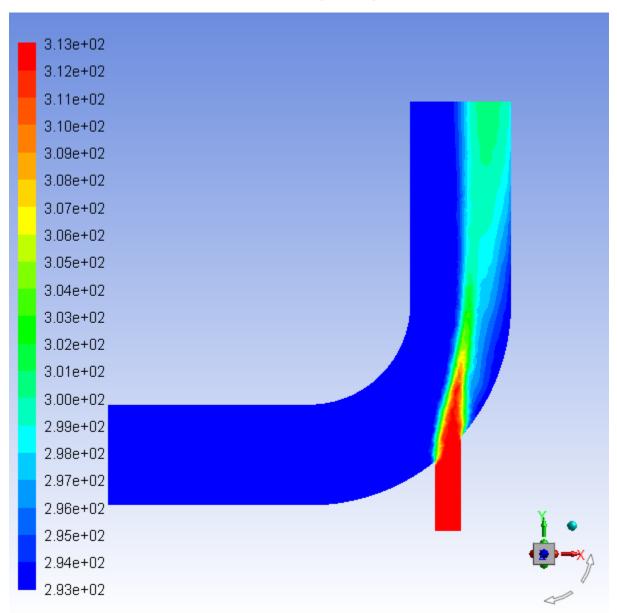


Figure 1.18: Temperature Distribution Along Symmetry Plane

c. Close the ANSYS Fluent application.

### File → Close Fluent

#### Important

Note that the ANSYS Fluent case and data files are automatically saved when you exit ANSYS Fluent and return to ANSYS Workbench.

d. View the list of files generated by ANSYS Workbench.

#### $View \rightarrow Files$

Introduction to Using ANSYS Fluent in ANSYS Workbench: Fluid Flow and Heat Transfer in a Mixing Elbow

Note the addition of the compressed ANSYS Fluent case and data files to the list of files. These will have names like FFF-1.cas.gz and FFF-1-00222.dat.gz. Note that the digit(s) following FFF may be different if you have had to restart the meshing or calculation steps for any reason and that the name of the data file is based on the number of iterations. Thus your file names may be slightly different than those shown here.

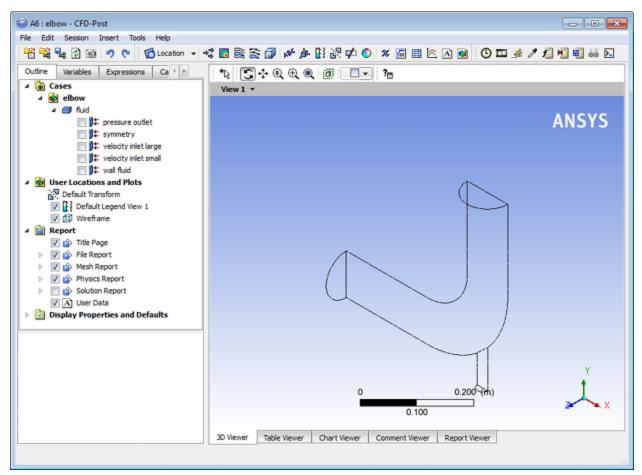
- 2. Display results in CFD-Post.
  - a. Start CFD-Post.

In the ANSYS Workbench **Project Schematic**, double-click the **Results** cell in the elbow fluid flow analysis system (cell A6). This displays the CFD-Post application. You can also right-click the **Results** cell to display the context menu where you can select the **Edit...** option.

#### Note

The elbow geometry is already loaded and is displayed in outline mode. ANSYS Fluent case and data files are also automatically loaded into CFD-Post.





b. Reorient the display.

Click the blue Z axis on the axis triad in the bottom right hand corner of the graphics display to orient the display so that the view is of the front of the elbow geometry.

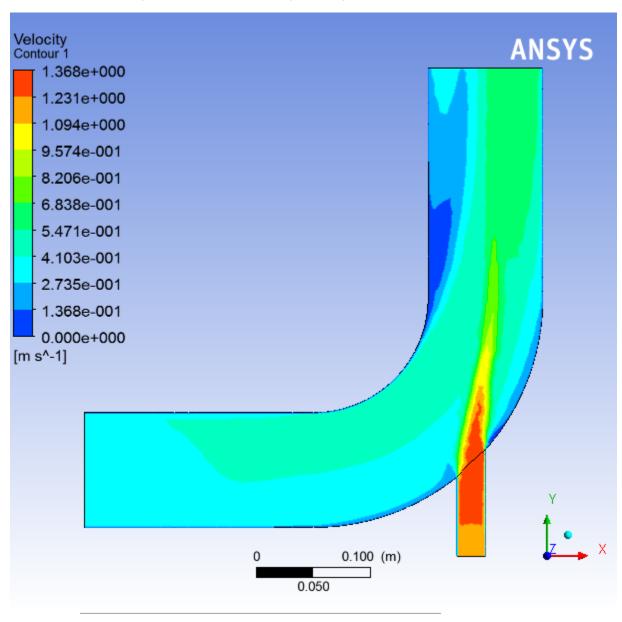
- c. Ensure that *Highlighting* ( <sup>(D)</sup>) is disabled.
- d. Display contours of velocity magnitude on the symmetry plane (Figure 1.20: Velocity Distribution Along Symmetry Plane (p. 58)).
  - i. Insert a contour object using the **Insert** menu item at the top of the CFD-Post window.

Insert  $\rightarrow$  Contour

This displays the Insert Contour dialog box.

- ii. Keep the default name of the contour (Contour 1) and click **OK** to close the dialog box. This displays the **Details of Contour 1** view below the **Outline** view in CFD-Post. This view contains all of the settings for a contour object.
- iii. In the Geometry tab, from the Domains drop-down list, select fluid.
- iv. In the Locations list, select symmetry.
- v. In the Variable list, select Velocity.
- vi. Click Apply.

Details of <b>Contour 1</b>								
Geometry	Labels Re	ender	View					
Domains	fluid				•			
Locations	symmetry				•			
Variable	Velocity				•	Ξ		
Range	Global				•			
Min				0 (m s^	-1]			
Max			1.3	86322 (m s^	-1]			
Boundary Data	а 🔘 н	ybrid	(	Onserv	/ative	-		
Apply				Reset	Defaul	ts		



#### Figure 1.20: Velocity Distribution Along Symmetry Plane

- e. Display contours of temperature on the symmetry plane (Figure 1.21: Temperature Distribution Along Symmetry Plane (p. 59)).
  - i. In the Outline tree view, under User Locations and Plots, clear the check box beside the Contour
     1 object to disable the Contour 1 object and hide the first contour display.
  - ii. Insert a contour object.

#### $\textbf{Insert} \rightarrow \textbf{Contour}$

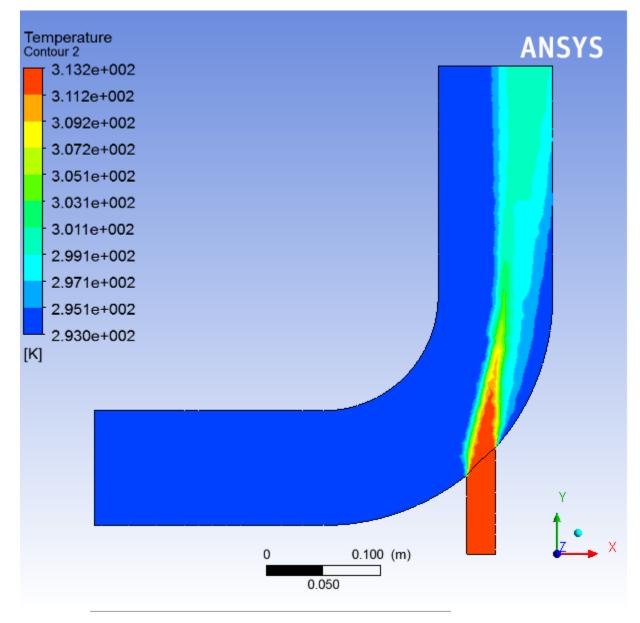
This displays the Insert Contour dialog box.

- iii. Keep the default name of the contour (Contour 2) and click **OK** to close the dialog box. This displays the **Details of Contour 2** view below the **Outline** view.
- iv. In the Geometry tab, from the Domains drop-down list, select fluid.

- v. In the Locations list, select symmetry.
- vi. In the Variable list, select Temperature.

vii. Click Apply.





3. Close the CFD-Post application by selecting **File** → **Close CFD-Post** or by clicking the 'X' in the top right corner of the window.

#### Important

Note that the CFD-Post state files are automatically saved when you exit CFD-Post and return to ANSYS Workbench.

Introduction to Using ANSYS Fluent in ANSYS Workbench: Fluid Flow and Heat Transfer in a Mixing Elbow

- 4. Save the elbow-workbench project in ANSYS Workbench.
- 5. View the list of files generated by ANSYS Workbench.

#### $\textbf{View} \rightarrow \textbf{Files}$

Note the addition of the CFD-Post state file (elbow.cst) to the list of files. For more information about CFD-Post (and the files associated with it), see the CFD-Post documentation.

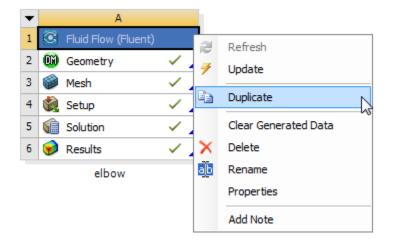
## 1.4.8. Duplicating the Fluent-Based Fluid Flow Analysis System

At this point, you have a completely defined fluid flow system that has a geometry, a computational mesh, a CFD setup and solution, and corresponding results. In order to study the effects upon the flow field that may occur if you were to alter the geometry, another fluid flow analysis is required. One approach would be to use the current system and change the geometry, however you would overwrite the data from your previous simulation. A more suitable and effective approach would be to create a copy, or duplicate, of the current system, and then make the appropriate changes to the duplicate system.

In this step, you will create a duplicate of the original Fluent-based fluid flow system, then review the list of files generated by ANSYS Workbench.

1. In the **Project Schematic**, right-click the title cell of the **Fluid Flow (Fluent)** system and select **Duplicate** from the context menu.

Figure 1.22: Duplicating the Fluid Flow System



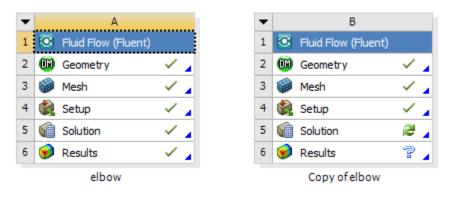


Figure 1.23: The Original Fluid Flow System and Its Duplicate

#### Note

Notice that in the duplicated system, the state of the **Solution** cell indicates that the cell requires a refresh while the state of the **Results** cell indicates that the cell requires attention. This is because when a system is duplicated, the case and data files are not copied to the new system, therefore, the new system does not yet have solution data associated with it.

- 2. Rename the duplicated system to new-elbow.
- 3. Save the elbow-workbench project in ANSYS Workbench.

### 1.4.9. Changing the Geometry in ANSYS DesignModeler

Now that you have two separate, but equivalent, Fluent-based fluid flow systems to work from, you can make changes to the second system without impacting the original system. In this step, you will make a slight alteration to the elbow geometry in ANSYS DesignModeler by changing the diameter of the smaller inlet, then review the list of files generated by ANSYS Workbench.

1. Open ANSYS DesignModeler.

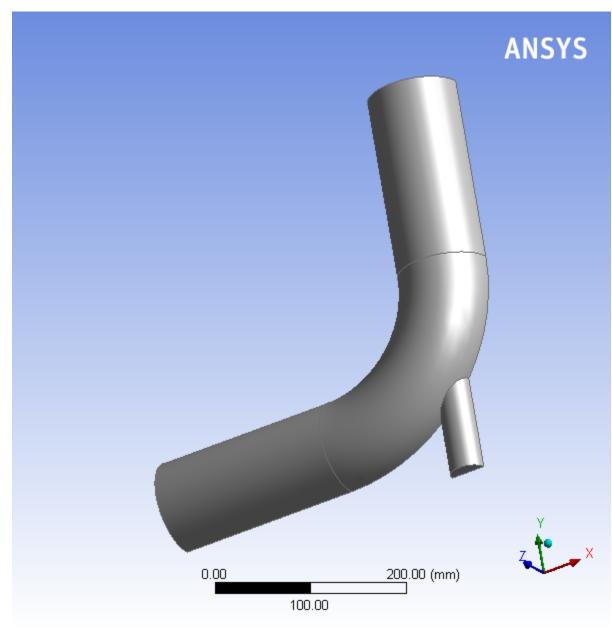
Right-click the **Geometry** cell of the new-elbow system (cell B2) and select **Edit Geometry in DesignModeler...** from the context menu to display the geometry in ANSYS DesignModeler.

- 2. Change the diameter of the small inlet (velocity-inlet-small).
  - a. In the Tree Outline, select Cylinder1 to open the Details View of the small inlet pipe.
  - b. In the Details View, change the FD10, Radius (>0) value from 12.5 millimeters to 19 millimeters.

Introduction to Using ANSYS Fluent in ANSYS Workbench: Fluid Flow and Heat Transfer in a Mixing Elbow

Details View						
Details of Cylinder1						
Cylinder	Cylinder1					
Base Plane	XYPlane					
Operation	Add Material					
Origin Definition	Coordinates					
FD3, Origin X Coordinate	137.5 mm					
FD4, Origin Y Coordinate	-225 mm					
FD5, Origin Z Coordinate	0 mm					
Axis Definition	Components					
FD6, Axis X Component	0 mm					
FD7, Axis Y Component	125 mm					
FD8, Axis Z Component	0 mm					
FD10, Radius (>0)	19 mm					
As Thin/Surface?	No					

c. Click the **Generate** button to generate the geometry with your new values.



#### Figure 1.24: Changing the Diameter of the Small Inlet in ANSYS DesignModeler

- 3. Close ANSYS DesignModeler.
- 4. View the list of files generated by ANSYS Workbench.

#### $\textbf{View} \rightarrow \textbf{Files}$

Note the addition of the geometry, mesh, and ANSYS Fluent settings files now associated with the new, duplicated system.

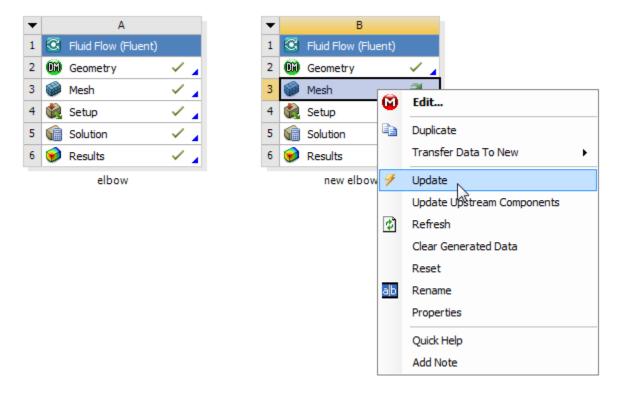
# 1.4.10. Updating the Mesh in the ANSYS Meshing Application

The modified geometry now requires a new computational mesh. The mesh settings for the duplicated system are retained in the duplicated system. In this step, you will update the mesh based on the mesh settings from the original system, then review the list of files generated by ANSYS Workbench.

Introduction to Using ANSYS Fluent in ANSYS Workbench: Fluid Flow and Heat Transfer in a Mixing Elbow

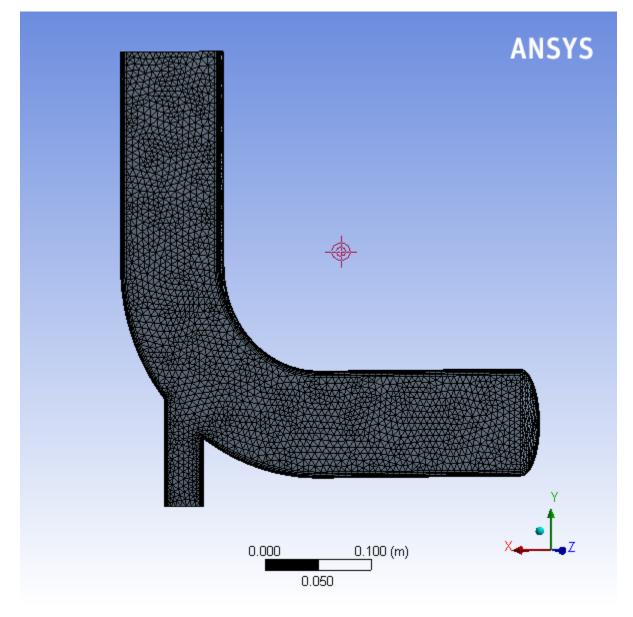
In the **Project Schematic**, right-click the **Mesh** cell of the new-elbow system (cell B3) and select **Update** from the context menu. This will update the mesh for the new geometry based on the mesh settings you specified earlier in the ANSYS Meshing application without having to open the ANSYS Meshing editor to regenerate the mesh.





It will take a few moments to update the mesh. Once the update is complete, the state of the **Mesh** cell is changed to up-to-date, symbolized by a green check mark.

For illustrative purposes of the tutorial, the new geometry and the new mesh are displayed below.



#### Figure 1.26: The Updated Geometry and Mesh in the ANSYS Meshing Application

Inspecting the files generated by ANSYS Workbench reveals the updated mesh file for the duplicated system.

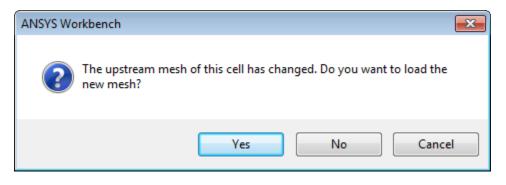
#### $View \rightarrow Files$

### 1.4.11. Calculating a New Solution in ANSYS Fluent

Now that there is an updated computational mesh for the modified geometry in the duplicated system, a new solution must be generated using ANSYS Fluent. In this step, you will revisit the settings within ANSYS Fluent, calculate another solution, view the new results, then review the list of files generated by ANSYS Workbench.

1. Open ANSYS Fluent.

In the **Project Schematic**, right-click the **Setup** cell of the new-elbow system (cell B4) and select **Edit...** from the context menu. Since the mesh has been changed, you are prompted as to whether you want to load the new mesh into ANSYS Fluent or not.



Click Yes to continue, and click OK when Fluent Launcher is displayed in order to open ANSYS Fluent.

2. Ensure that the unit of length is set to millimeters.



3. Check the mesh (optional).

```
Setting Up Domain \rightarrow Mesh \rightarrow Check
```

4. Revisit the boundary conditions for the small inlet.



In the **Velocity Inlet** dialog box for **velocity-inlet-small**, you must set the hydraulic diameter to 38 mm based on the new dimensions of the small inlet.

5. Re-initialize the solution.

Solving → Initialization

Keep the **Method** at the default of **Hybrid** and click **Initialize**.

6. Recalculate the solution.

Solving  $\rightarrow$  Run Calculation  $\rightarrow$  Calculate

Keep the No. of Iterations set to 300 and click Calculate.

- 7. Close ANSYS Fluent.
- 8. Revisit the results of the calculations in CFD-Post.

Double-click the **Results** cell of the new-elbow fluid flow system to re-open CFD-Post where you can review the results of the new solution.

- 9. Close CFD-Post.
- 10. Save the elbow-workbench project in ANSYS Workbench.

11. View the list of files generated by ANSYS Workbench.

#### $View \rightarrow Files$

Note the addition of the solution and state files now associated with new duplicated system.

### **1.4.12.** Comparing the Results of Both Systems in CFD-Post

In this step, you will create a new **Results** system in ANSYS Workbench, use that system to compare the solutions from each of the two Fluent-based fluid flow analysis systems in CFD-Post at the same time, then review the list of files generated by ANSYS Workbench.

1. Create a **Results** system.

In ANSYS Workbench, drag a **Results** system from the **Component Systems** section of the **Toolbox** and drop it into the **Project Schematic**, next to the fluid flow systems.

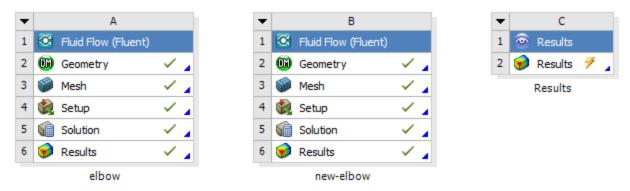
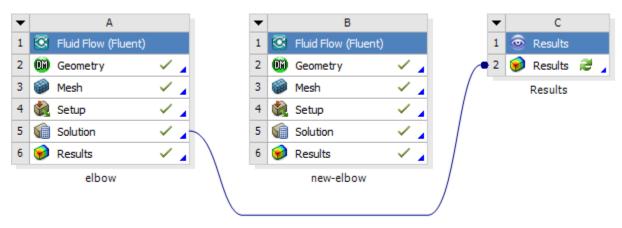


Figure 1.27: The NewResults System in the Project Schematic

- 2. Add the solutions of each of the systems to the **Results** system.
  - a. Select the **Solution** cell in the first Fluid Flow analysis system (cell A5) and drag it over the **Results** cell in the **Results** system (cell C2). This creates a transfer data connection between the two systems.

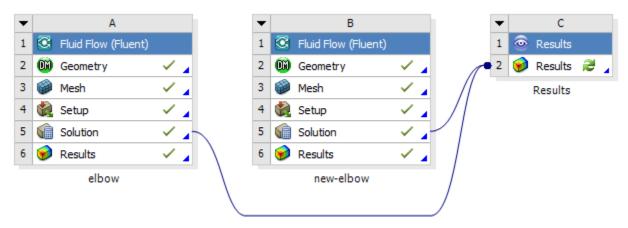
Figure 1.28: Connecting the First Fluid Flow System to the New Results System



Introduction to Using ANSYS Fluent in ANSYS Workbench: Fluid Flow and Heat Transfer in a Mixing Elbow

b. Select the **Solution** cell in the second Fluid Flow analysis system (cell B5) and drag it over the **Results** cell in the **Results** system (cell C2). This creates a transfer data connection between the two systems.

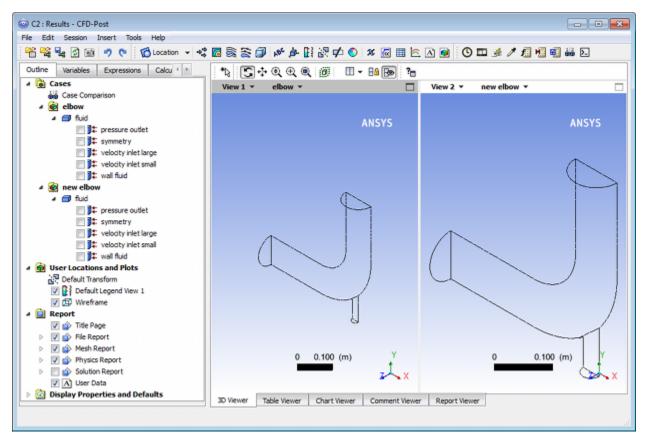
Figure 1.29: Connecting the Second Fluid Flow System to the New Results System



3. Open CFD-Post to compare the results of the two fluid flow systems.

Now that the two fluid flow systems are connected to the **Results** system, double-click the **Results** cell in the **Results** system (cell C2) to open CFD-Post. Within CFD-Post, both geometries are displayed side by side.





a. Re-orient the display.

In each view, click the blue Z axis on the axis triad in the bottom right hand corner of the graphics display to orient the display so that the view is of the front of the elbow geometry.

#### Important

Alternatively, you can select the synchronization tool ( **D**) in the **3D Viewer Toolbar** to synchronize the views, so that when you re-orient one view, the other view is automatically updated.

- b. Display contours of velocity magnitude on the symmetry plane.
  - i. Insert a contour object.

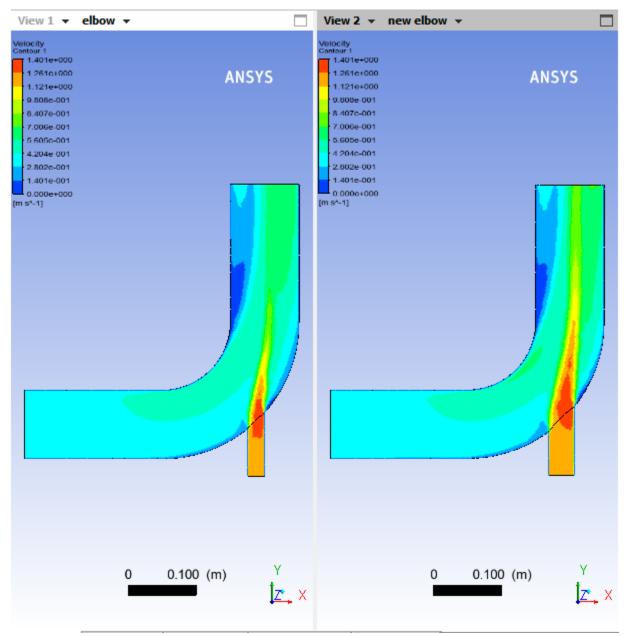
#### $\textbf{Insert} \rightarrow \textbf{Contour}$

This displays the Insert Contour dialog box.

- ii. Keep the default name of the contour (Contour 1) and click OK to close the dialog box. This displays the Details of Contour 1 view below the Outline view in CFD-Post. This view contains all of the settings for a contour object.
- iii. In the Geometry tab, from the Domains list, select fluid.
- iv. In the Locations list, select symmetry.
- v. In the Variable list, select Velocity.
- vi. Click **Apply**. The velocity contours are displayed in each view.

#### Note

To better visualize the velocity display, you can clear the **Wireframe** view option under **User Locations and Plots** in the **Outline** tree view.





- c. Display contours of temperature on the symmetry plane.
  - i. In the **Outline** tree view, under **User Locations and Plots**, deselect the **Contour 1** object to hide the first contour display in CFD-Post.
  - ii. Insert another contour object.

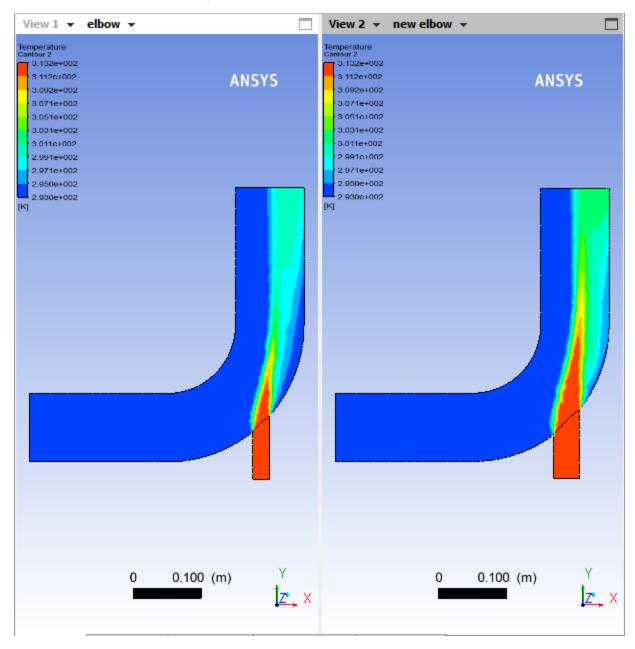
### $\textbf{Insert} \rightarrow \textbf{Contour}$

This displays the Insert Contour dialog box.

iii. Keep the default name of the contour (Contour 2) and click **OK** to close the dialog box. This displays the **Details of Contour 2** view below the **Outline** view in CFD-Post.

- iv. In the Geometry tab, select fluid in the Domains list.
- v. Select symmetry in the Locations list.
- vi. Select Temperature in the Variable list.
- vii. Click **Apply**. The temperature contours are displayed in each view.

Figure 1.32: CFD-Post Displaying Temperature Contours for Both Geometries



- 4. Close the CFD-Post application.
- 5. Save the elbow-workbench project in ANSYS Workbench.
- 6. View the list of files associated with your project using the **Files** view.

#### $View \rightarrow Files$

Note the addition of the **Results** system and its corresponding files.

### 1.5. Summary

In this tutorial, portions of ANSYS Workbench were used to compare the fluid flow through two slightly different geometries. ANSYS DesignModeler was used to create a mixing elbow geometry, ANSYS Meshing was used to create a computational mesh, ANSYS Fluent was used to calculate the fluid flow throughout the geometry using the computational mesh, and CFD-Post was used to analyze the results. In addition, the geometry was altered, a new mesh was generated, and a new solution was calculated. Finally, ANSYS Workbench was set up so that CFD-Post could directly compare the results of both calculations at the same time.

# Chapter 2: Parametric Analysis in ANSYS Workbench Using ANSYS Fluent

This tutorial is divided into the following sections:

- 2.1. Introduction
- 2.2. Prerequisites
- 2.3. Problem Description
- 2.4. Setup and Solution

# 2.1. Introduction

This tutorial illustrates using an ANSYS Fluent fluid flow system in ANSYS Workbench to set up and solve a three-dimensional turbulent fluid flow and heat transfer problem in an automotive heating, ventilation, and air conditioning (HVAC) duct system. ANSYS Workbench uses parameters and design points to allow you to run optimization and what-if scenarios. You can define both input and output parameters in ANSYS Fluent that can be used in your ANSYS Workbench project. You can also define parameters in other applications including ANSYS DesignModeler and ANSYS CFD-Post. Once you have defined parameters for your system, a **Parameters** cell is added to the system and the **Parameter Set** bus bar is added to your project. This tutorial is designed to introduce you to the parametric analysis utility available in ANSYS Workbench.

The tutorial starts with a Fluid Flow (Fluent) analysis system with pre-defined geometry and mesh components. Within this tutorial, you will redefine the geometry parameters created in ANSYS Design-Modeler by adding constraints to the input parameters. You will use ANSYS Fluent to set up and solve the CFD problem. While defining the problem set-up, you will also learn to define input parameters in ANSYS Fluent. The tutorial will also provide information on how to create output parameters in ANSYS CFD-Post.

This tutorial demonstrates how to do the following:

- Add constraints to the ANSYS DesignModeler input parameters.
- Create an ANSYS Fluent fluid flow analysis system in ANSYS Workbench.
- Set up the CFD simulation in ANSYS Fluent, which includes:
  - Setting material properties and boundary conditions for a turbulent forced convection problem.
  - Defining input parameters in Fluent
- Define output parameters in CFD-Post
- Create additional design points in ANSYS Workbench.
- Run multiple CFD simulations by updating the design points.

• Analyze the results of each design point project in ANSYS CFD-Post and ANSYS Workbench.

#### Important

The mesh and solution settings for this tutorial are designed to demonstrate a basic parameterization simulation within a reasonable solution time-frame. Ordinarily, you would use additional mesh and solution settings to obtain a more accurate solution.

### 2.2. Prerequisites

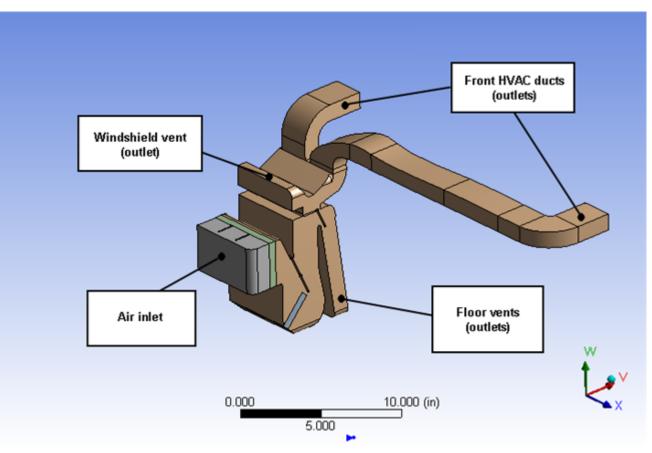
This tutorial assumes that you are already familiar with the ANSYS Workbench interface and its project workflow (for example, ANSYS DesignModeler, ANSYS Meshing, ANSYS Fluent, and ANSYS CFD-Post). This tutorial also assumes that you have completed Introduction to Using ANSYS Fluent in ANSYS Workbench: Fluid Flow and Heat Transfer in a Mixing Elbow (p. 1), and that you are familiar with the ANSYS Fluent graphical user interface. Some steps in the setup and solution procedure will not be shown explicitly.

# 2.3. Problem Description

In the past, evaluation of vehicle air conditioning systems was performed using prototypes and testing their performance in test labs. However, the design process of modern vehicle air conditioning (AC) systems improved with the introduction of Computer Aided Design (CAD), Computer Aided Engineering (CAE) and Computer Aided Manufacturing (CAM). The AC system specification will include minimum performance requirements, temperatures, control zones, flow rates, and so on. Performance testing using CFD may include fluid velocity (air flow), pressure values, and temperature distribution. Using CFD enables the analysis of fluid through very complex geometry and boundary conditions.

As part of the analysis, a designer can change the geometry of the system or the boundary conditions such as the inlet velocity, flow rate, and so on, and view the effect on fluid flow patterns. This tutorial illustrates the AC design process on a representative automotive HVAC system consisting of both an evaporator for cooling and a heat exchanger for heating requirements. This HVAC system is symmetric, so the geometry has been simplified using a plane of symmetry to reduce computation time.





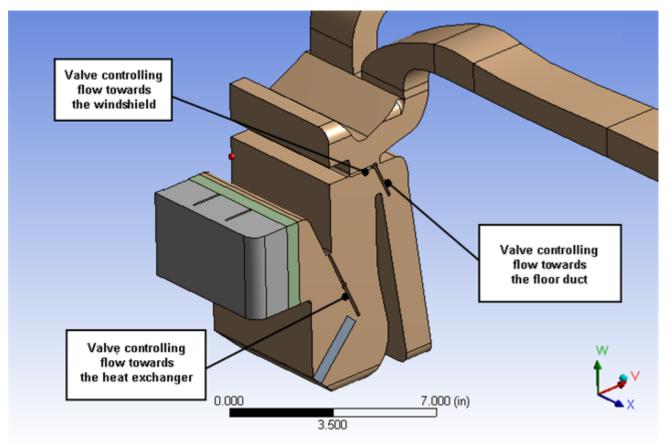


Figure 2.2: HVAC System Valve Location Details

Figure 2.1: Automotive HVAC System (p. 75) shows a representative automotive HVAC system. The system has three valves (as shown in Figure 2.2: HVAC System Valve Location Details (p. 76)), which control the flow in the HVAC system. The three valves control:

- Flow over the heat exchanger coils
- Flow towards the duct controlling the flow through the floor vents
- Flow towards the front vents or towards the windshield

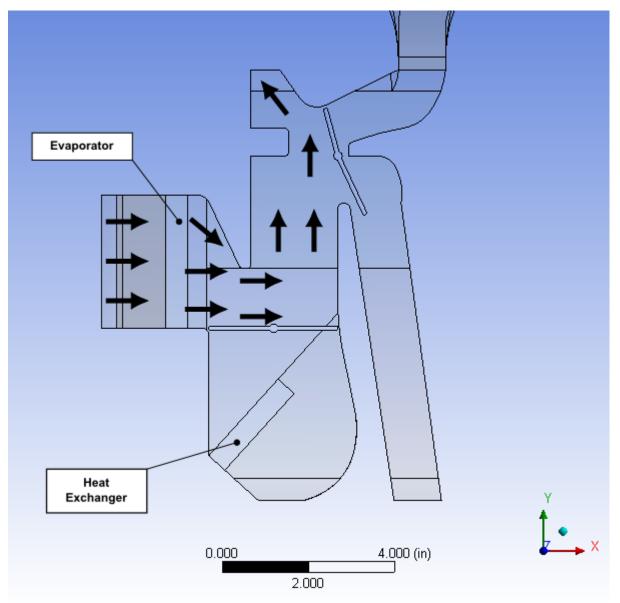
Air enters the HVAC system at 310 K with a velocity of 0.5 m/sec through the air inlet and passes to the evaporator and then, depending on the position of the valve controlling flow to the heat exchanger, flows over or bypasses the heat exchanger. Depending on the cooling and heating requirements, either the evaporator or the heat exchanger would be operational, but not both at the same time. The position of the other two valves controls the flow towards the front panel, the windshield, or towards the floor ducts.

The motion of the valves is constrained. The valve controlling flow over the heat exchanger varies between 25° and 90°. The valve controlling the floor flow varies between 20° and 60°. The valve controlling flow towards front panel or windshield varies between 15° and 175°.

The evaporator load is about 200 W in the cooling cycle. The heat exchanger load is about 150 W.

This tutorial illustrates the easiest way to analyze the effects of the above parameters on the flow pattern/distribution and the outlet temperature of air (entering the passenger cabin). Using the parametric analysis capability in ANSYS Workbench, a designer can check the performance of the system at various design points.





# 2.4. Setup and Solution

To help you quickly identify graphical user interface items at a glance and guide you through the steps of setting up and running your simulation, the ANSYS Fluent Tutorial Guide uses several type styles and mini flow charts. See Typographical Conventions Used In This Manual (p. xvi) for detailed information.

The following sections describe the setup and solution steps for this tutorial:

2.4.1. Preparation
2.4.2. Adding Constraints to ANSYS DesignModeler Parameters in ANSYS Workbench
2.4.3. Setting Up the CFD Simulation in ANSYS Fluent
2.4.4. Defining Input Parameters in ANSYS Fluent
2.4.5. Solving

2.4.6. Postprocessing and Setting the Output Parameters in ANSYS CFD-Post

2.4.7. Creating Additional Design Points in ANSYS Workbench 2.4.8. Postprocessing the New Design Points in CFD-Post 2.4.9. Summary

## 2.4.1. Preparation

To prepare for running this tutorial:

- 1. Set up a working folder on the computer you will be using.
- 2. Go to the ANSYS Customer Portal, https://support.ansys.com/training.

#### Note

If you do not have a login, you can request one by clicking **Customer Registration** on the log in page.

- 3. Enter the name of this tutorial into the search bar.
- 4. Narrow the results by using the filter on the left side of the page.
  - a. Click ANSYS Fluent under Product.
  - b. Click 18.0 under Version.
- 5. Select this tutorial from the list.
- 6. Click the **workbench-parameter-tutorial\_R180.zip** link to download the input files.
- 7. Unzip the workbench-parameter-tutorial\_R180.zip file to your working folder.

The extracted workbench-parameter-tutorial folder contains a single archive file fluentworkbench-param.wbpz that includes all supporting input files of the starting ANSYS Workbench project and a folder called final\_project\_files that includes the archived final version of the project. The final result files incorporate ANSYS Fluent and ANSYS CFD-Post settings and all already defined design points (all that is required is to update the design points in the project to generate corresponding solutions).

#### Note

ANSYS Fluent tutorials are prepared using ANSYS Fluent on a Windows system. The screen shots and graphic images in the tutorials may be slightly different than the appearance on your system, depending on the operating system or graphics card.

### 2.4.2. Adding Constraints to ANSYS DesignModeler Parameters in ANSYS Workbench

In this step, you will start ANSYS Workbench, open the project file, review existing parameters, create new parameters, and add constraints to existing ANSYS DesignModeler parameters.

1. From the Windows Start menu, select Start > All Programs > ANSYS 18.0 > Workbench 18.0 to start ANSYS Workbench.

This displays the ANSYS Workbench application window, which has the **Toolbox** on the left and the **Project Schematic** to its right. Various supported applications are listed in the **Toolbox**, and the components of the analysis system are displayed in the **Project Schematic**.

#### Note

When you first start ANSYS Workbench, the **Getting Started** message window is displayed, offering assistance through the online help for using the application. You can keep the window open, or close it by clicking **OK**. If you need to access the online help at any time, use the **Help** menu, or press the **F1** key.

2. Restore the archive of the starting ANSYS Workbench project to your working directory.

#### File $\rightarrow$ Restore Archive...

The Select Archive to Restore dialog box appears.

a. Browse to your working directory, select the project archive file fluent-workbench-param.wbpz, and click **Open**.

The Save As dialog box appears.

b. Browse, if necessary, to your working folder and click **Save** to restore the project file, fluent-workbench-param.wbpj, and a corresponding project folder, fluent-workbench-param\_files, for this tutorial.

Now that the project archive has been restored, the project will automatically open in ANSYS Workbench.

<b>A</b>	
A fluent-workbench-param - Workbench	
File View Tools Units Extensions J	obs Help
Project	
👔 Import 🍦 Reconnect 😰 Refresh Projec	t 🐬 Update Project 🐬 Update All Design Points 📲 ACT Start Pa
Toolbox 🝷 🕂 🗙	Project Schematic
□ Analysis Systems	
🔇 Fluid Flow (Fluent)	
IC Engine (Fluent)	▼ A
🚽 Throughflow	1 S Fluid Flow (FLUENT)
Hroughflow (BladeGen)	2 🕅 Geometry 🗸
Component Systems	3 🍘 Mesh 🛛 🥰 🖌
	4 🍓 Setup 😨
External Connection Systems	5 🖬 Solution 💡
	6 😡 Results 😨
	> 7 Parameters
	Fluid Flow (FLUENT)
	Parameter Set
View All / Customize	
🔋 Ready	Job Monitor 💷 Show Progress 🔑 S

Figure 2.4: The Project Loaded into ANSYS Workbench

The project (fluent-workbench-param.wbpj) already has a Fluent-based fluid flow analysis system that includes the geometry and mesh, as well as some predefined parameters. You will first examine and edit parameters within Workbench, then later proceed to define the fluid flow model in ANSYS Fluent.

3. Open the **Files** view in ANSYS Workbench so you can view the files associated with the current project and are written during the session.

 $\textbf{View} \rightarrow \textbf{Files}$ 

N fluent-workbench-param - Workbench File View Tools Units Extensions	Jobs Help							
	5005 1100							
Import	_		Design Points		ACT Start	'age		
Toolbox 🔻 🖡 🗙	Project Sch	ematic						
Analysis Systems								
<ul> <li>Fluid Flow (Fluent)</li> <li>IC Engine (Fluent)</li> </ul>	-	A						
Throughflow	1	Fluid Flow (FLUENT						
Throughflow (BladeGen)	2	Geometry						
Component Systems	3	Mesh	2					
Design Exploration	4	40	7					
External Connection Systems	5	Setup	7					
		Solution						
	6	Results	? .					
	7	Parameters						
	िंग्रे Para	ameter Set						
	්ට Para	ameter Set						
		ameter Set						
	िंग्न Para	ameter Set			B	с	D	
			_		B Ce 💌	C Size V	D	•
	Files	A	am.wbpj	•	-	-	_	
	Files	A Name	am.wbpj	<b>•</b>	-	Size 💌	Туре	oject File
	Files	A Name fluent-workbench-par	am.wbpj	•	Ce 💌	Size 134 KB	Type Workbench Pr	oject File
	Files	A Name fluent-workbench-par Geom.agdb FFF.agdb FFF.mshdb	am.wbpj		Ce 💌	Size ▼ 134 K8 2 M8 2 M8 184 K8	Type Workbench Pr Geometry File Geometry File .mshdb	oject File
	Files	A Name fluent-workbench-par Geom.agdb FFF.agdb FFF.mshdb designPoint.wbdp	am.wbpj		Ce	Size	Type Workbench Pri Geometry File .mshdb Workbench De	oject File
	Files	A Name fluent-workbench-par Geom.agdb FFF.agdb FFF.mshdb	am.wbpj		Ce	Size ▼ 134 K8 2 M8 2 M8 184 K8	Type Workbench Pr Geometry File Geometry File .mshdb	oject File
	Files	A Name fluent-workbench-par Geom.agdb FFF.agdb FFF.mshdb designPoint.wbdp	am.wbpj		Ce	Size	Type Workbench Pri Geometry File .mshdb Workbench De	oject File
	Files	A Name fluent-workbench-par Geom.agdb FFF.agdb FFF.mshdb designPoint.wbdp	am.wbpj		Ce	Size	Type Workbench Pri Geometry File .mshdb Workbench De	oject File
▼ View AI / Customize	Files	A Name fluent-workbench-par Geom.agdb FFF.agdb FFF.agdb FFF.mshdb designPoint.wbdp DesignPointLog.csv	am.wbpj		Ce	Size	Type Workbench Pri Geometry File .mshdb Workbench De	oject File

Figure 2.5: The Project Loaded into ANSYS Workbench Displaying Properties and Files View

Note the types of files that have been created for this project. Also note the states of the cells for the Fluid Flow (Fluent) analysis system. Since the geometry has already been defined, the status of the **Geometry** cell is Up-to-Date ( $\checkmark$ ). Since the mesh is not complete, the **Mesh** cell's state is Refresh Required ( $\gtrless$ ), and since the ANSYS Fluent setup is incomplete and the simulation has yet to be performed, with

no corresponding results, the state for the **Setup**, **Solution**, and **Results** cells is Unfulfilled ( $\mathbb{P}$ ). For more information about cell states, see the Workbench User's Guide.

- 4. Review the input parameters that have already been defined in ANSYS DesignModeler.
  - a. Double-click the **Parameter Set** bus bar in the ANSYS Workbench **Project Schematic** to open the **Parameters Set** tab.

Note

To return to viewing the **Project Schematic**, click the **Project** tab.

- b. In the **Outline of All Parameters** view (Figure 2.6: Parameters Defined in ANSYS DesignModeler (p. 82)), review the following existing parameters:
  - The parameter hcpos represents the valve position that controls the flow over the heat exchanger. When the valve is at an angle of 25°, it allows the flow to pass over the heat exchanger. When the angle is 90°, it completely blocks the flow towards the heat exchanger. Any value in between allows some flow to pass over the heat exchanger giving a mixed flow condition.
  - The parameter ftpos represents the valve position that controls flow towards the floor duct. When the valve is at an angle of 20°, it blocks the flow towards the floor duct and when the valve angle is 60°, it unblocks the flow.
  - The parameter wsfpos represents the valve position that controls flow towards the windshield and the front panel. When the valve is at an angle of 15°, it allows the entire flow to go towards the windshield. When the angle is 90°, it completely blocks the flow towards windshield as well as the front panel. When the angle is 175°, it allows the flow to go towards the windshield and the front panel.

	А	В	с	D
1	ID	Parameter Name	Value	Unit
2	Input Parameters			
3	🗏 🖾 Fluid Flow (FLUENT) (A1)			
4	ί <mark>ρ</mark> Ρ1	hcpos	90	
5	ί <mark>ρ</mark> Ρ2	ftpos	25	
6	🗘 РЗ	wsfpos	175	
*	🗘 New input parameter	New name	New expression	
8	Output Parameters			
*	🔁 New output parameter		New expression	
10	Charts			

#### Figure 2.6: Parameters Defined in ANSYS DesignModeler

- 5. In the **Outline of All Parameters** view, create three new named input parameters.
  - a. In the row that contains **New input parameter**, click the parameter table cell with **New name** (under the **Parameter Name** column) and enter **input\_hcpos**. Note the ID of the parameter that appears in column **A** of the table. For the new input parameter, the parameter **ID** is **P4**. In the **Value** column, enter **15**.
  - b. In a similar manner, create two more parameters named input\_ftpos and input\_wsfpos. In the Value column, enter 25, and 90 for each new parameter (P5 and P6), respectively.

	А	В	с	D
1	ID	Parameter Name	Value	Unit
2	Input Parameters			
3	🖃 区 Fluid Flow (FLUENT) (A1)			
4	ι <mark>ρ</mark> Ρ1	hcpos	90	
5	ι <mark>ρ</mark> Ρ2	ftpos	25	
6	🗘 РЗ	wsfpos	175	
7	🗘 Р4	input_hcpos	15	
8	🗘 Р5	input_ftpos	25	
9	🗘 Рб	input_wsfpos	90	
*	🏟 New input parameter	New name	New expression	
11	Output Parameters			
*	🔁 New output parameter		New expression	
13	Charts			

#### Figure 2.7: New Parameters Defined in ANSYS Workbench

6. Select the row (or any cell in the row) that corresponds to the hcpos parameter. In the Properties of Outline view, change the value of the hcpos parameter in the Expression field from 90 to the expression min(max(25,P4),90). This puts a constraint on the value of hcpos, so that the value always remains between 25° and 90°. The redefined parameter hcpos is automatically passed to ANSYS DesignModeler. Alternatively the same constraint can also be set using the expression max(25, min(P4,90)).

After defining this expression, the parameter becomes a derived parameter that is dependent on the value of the parameter input\_hcpos with ID **P4**. The derived parameters are unavailable for editing in the **Outline of All Parameters** view and could be redefined only in the **Properties of Outline** view.

#### Important

When entering expressions, you must use the list and decimal delimiters associated with your selected language in the Workbench regional and language settings, which correspond to the regional settings on your machine. The instructions in this tutorial assume that your systems uses "." as a decimal separator and "," as a list separator.

Figure 2.8:	Constrained	<b>Parameter</b> hcpos
-------------	-------------	------------------------

Outline	Outline of All Parameters 🔹 📮 🗙						
	А		В	с			
1	ID		Parameter Name	Value			
2	Input Parameters						
3	🖃 🙆 Fluid Flow (FLU	IENT) (A1)					
4	ι <mark>ρ</mark> Ρ1		hcpos	25			
5	🛱 P2		ftpos	25			
6	🗘 P3		wsfpos	175			
7	ί <mark>ρ</mark> Ρ4		input_hcpos	15			
8	ί <mark>ρ</mark> Ρ5		input_ftpos	25			
9	🗘 P6		input_wsfpos	90			
*	🗘 New input para	ameter	New name	New expression			
11	Output Parameters						
*	😡 New output pa	rameter		New expression			
13	Charts						
•				4			
Properti	es of Outline A4: P1			<b>→</b> ∓ X			
	A		В				
1	Property		Value				
2	General						
3	Expression	min(max(2	5,P4),90)				
4	Description						
5	Error Message						
6	Expression Type	Derived					
7	Usage	Input					
8	Quantity Name	Dimension	ess				

 Select the row or any cell in the row that corresponds to the ftpos parameter and create a similar expression for ftpos:min(max(20, P5), 60).

#### Outline of All Parameters **ч** д х С A В ID Parameter Name Value 1 Input Parameters 2 Fluid Flow (FLUENT) (A1) 3 ίρ P1 4 hcpos 25 ..... 🕻 P2 5 ftpos 25 🕻 P3 6 wsfpos 175 🕻 P4 7 input\_hcpos 15 8 🕻 P5 input\_ftpos 25 ۲þ. 9 P6 input\_wsfpos 90 \* New input parameter New name New expression Output Parameters 11 -\* New output parameter New expression 13 Charts Ш < □ Þ Properties of Outline B5: P2 **⊸ д х** A в 1 Property Value General 2 3 Expression min(max(20,P5),60) 4 Description 5 Error Message 6 Expression Type Derived 7 Usage Input 8 Quantity Name Dimensionless

#### Figure 2.9: Constrained Parameterftpos

8. Create a similar expression for wsfpos:min(max(15,P6),175).

Outline of All Parameters 🔹 📮 🗙							
	А		В	с			
1	ID		Parameter Name	Value			
2	Input Parameters						
3	🖃 🙆 Fluid Flow (FLU	ENT) (A1)					
4	ι <mark>ρ</mark> Ρ1		hcpos	25			
5	ι <mark>φ</mark> Ρ2		ftpos	25			
6	ί <mark>ρ</mark> Ρ3		wsfpos	90			
7	ί <mark>ρ</mark> Ρ4		input_hcpos	15			
8	ί <mark>ρ</mark> Ρ5		input_ftpos	25			
9	ί <mark>ρ</mark> Ρ6		input_wsfpos	90			
*	🗘 New input para	ameter	New name	New expression			
11	Output Parameters						
*	Rew output pa	rameter		New expression			
13	Charts						
•				•			
	four compa						
Propertie	es of Outline C6: P3			~ д Х			
	A		В				
1	Property		Value				
2	General						
3	Expression	min(max(1	5,P6),175)				
4	Description						
5	Error Message						
6	Expression Type	Derived					
7	Usage	Input					
8	Quantity Name	Dimension	ess				

#### 9. Click the **X** on the right side of the **Parameters Set** tab to close it and return to the **Project Schematic**.

Note the new status of the cells in the Fluid Flow (Fluent) analysis system. Since we have changed the values of hcpos, ftpos, and wsfpos to their new expressions, the **Geometry** and **Mesh** cells now indicates Refresh Required ( $\gtrless$ ).

#### 10. Update the **Geometry** and **Mesh** cells.

- a. Right-click the **Geometry** cell and select the **Update** option from the context menu.
- b. Likewise, right-click the **Mesh** cell and select the **Refresh** option from the context menu. Once the cell is refreshed, then right-click the **Mesh** cell again and select the **Update** option from the context menu.
- 11. Save the project in ANSYS Workbench.

In the main menu, select File  $\rightarrow$  Save

## 2.4.3. Setting Up the CFD Simulation in ANSYS Fluent

Now that you have edited the parameters for the project, you will set up a CFD analysis using ANSYS Fluent. In this step, you will start ANSYS Fluent, and begin setting up the CFD simulation.

### 2.4.3.1. Starting ANSYS Fluent

In the ANSYS Workbench **Project Schematic**, double-click the **Setup** cell in the ANSYS Fluent fluid flow analysis system. You can also right-click the **Setup** cell to display the context menu where you can select the **Edit...** option.

When ANSYS Fluent is first started, Fluent Launcher is displayed, allowing you to view and/or set certain ANSYS Fluent start-up options.

Fluent Launcher allows you to decide which version of ANSYS Fluent you will use, based on your geometry and on your processing capabilities.

Figure 2.11: ANSYS Fluent Launcher

E Fluent Launcher (Setting Edit Only)						
<b>ANSYS</b>	Fluent Launcher					
Dimension © 2D @ 3D	Options Double Precision  Meshing Mode					
Display Options ☑ Display Mesh After Reading ☑ Workbench Color Scheme □ Do not show this panel again	Processing Options Serial  Parallel					
ACT Option						
Show More Options						
OK Cancel Help -						

1. Ensure that the proper options are enabled.

#### Important

Note that the **Dimension** setting is already filled in and cannot be changed, since ANSYS Fluent automatically sets it based on the mesh or geometry for the current system.

a. Ensure that the **Display Mesh After Reading** and **Workbench Color Scheme** options are enabled.

#### Note

An option is enabled when there is a check mark in the check box, and disabled when the check box is empty. To change an option from disabled to enabled (or vice versa), click the check box or the text.

b. Ensure that Serial is selected from the Processing Options list.

#### Note

Parallel processing offers a substantial reduction in computational time. Refer to Introduction to Using ANSYS Fluent: Fluid Flow and Heat Transfer in a Mixing Elbow (p. 121) in this manual and the Fluent User's Guide for further information about using the parallel processing capabilities of ANSYS Fluent.

c. Ensure that the **Double Precision** option is disabled.

#### Note

Fluent will retain your preferences for future sessions.

2. Click **OK** to launch ANSYS Fluent.

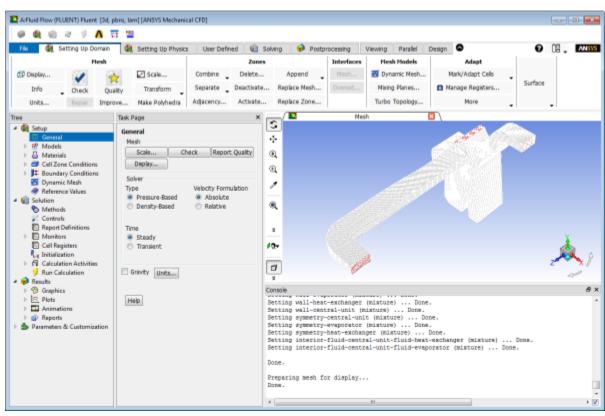


Figure 2.12: The ANSYS Fluent Application

### 2.4.3.2. Setting Up Physics

1. In the **Solver** group of the **Setting Up Physics** ribbon tab, retain the default selection of the steady pressurebased solver.

	Setting Up Physics → Solver							
	File 👹	Setting Up Domain	🍓 Setting Up I	Physics	User Defined 🧃			
			Solver					
	Time     Type       Image: Steady     Image: Pressure-Based       Image: Transient     Image: Density-Based		Velocity Formulation Output	Operati	ng Conditions			
			Relative	Reference Values				

2. Set up your models for the CFD simulation using the **Models** group of the **Setting Up Physics** ribbon tab.

Energy	Radiation	Nultiphase	🚨 Solidify/Melt		
	<sup>≫</sup> <sub>≠</sub> Heat Exchanger	🔿 Species	动) Acoustics		
	🕓 Viscous	📑 Discrete Phase	🗄 More 🖕		
Models					

a. Enable heat transfer by activating the energy equation.

In the Setting Up Physics ribbon tab, select Energy (Models group).



b. Enable the k- $\varepsilon$  turbulence model.

Setting Up Physics → Models → Viscous				
<b>U</b> iscous Model		<b>—</b> ×		
Model	Model Constants			
Inviscid	Cmu	<b>A</b>		
🔘 Laminar	0.09			
Spalart-Allmaras (1 eqn)	C1-Epsilon			
k-epsilon (2 eqn)	1.44			
<ul> <li>k-omega (2 eqn)</li> <li>Transition k-kl-omega (3 eqn)</li> </ul>	C2-Epsilon			
<ul> <li>Transition SST (4 eqn)</li> </ul>	1.92	=		
Reynolds Stress (7 eqn)	TKE Prandtl Number			
Scale-Adaptive Simulation (SAS)	1			
Detached Eddy Simulation (DES)	TDR Prandtl Number			
Carge Eddy Simulation (LES)	1.3			
k-epsilon Model	Energy Prandtl Number			
Standard	0.85			
RNG     Realizable	Wall Prandtl Number	Ŧ		
<ul> <li>Near-Wall Treatment</li> <li>Standard Wall Functions</li> <li>Scalable Wall Functions</li> <li>Scalable Wall Functions</li> <li>Non-Equilibrium Wall Functions</li> <li>Enhanced Wall Treatment</li> <li>Menter-Lechner</li> <li>User-Defined Wall Functions</li> <li>Enhanced Wall Treatment Options</li> <li>Pressure Gradient Effects</li> <li>Thermal Effects</li> <li>Options</li> <li>Viscous Heating</li> <li>Curvature Correction</li> <li>Production Kato-Launder</li> <li>Production Limiter</li> </ul>	User-Defined Functions Turbulent Viscosity none Prandtl Numbers TKE Prandtl Number none TDR Prandtl Number none Energy Prandtl Number none Wall Prandtl Number none	• • •		
OK Cancel Help				

- i. Select **k-epsilon (2 eqn)** from the **Model** group box.
- ii. Select Enhanced Wall Treatment from the Near-Wall Treatment group box.

The default Standard Wall Functions are generally applicable when the cell layer adjacent to the wall has a y+ larger than 30. In contrast, the Enhanced Wall Treatment option provides consistent solutions for all y+ values. Enhanced Wall Treatment is recommended when using the k-epsilon model for general single-phase fluid flow problems. For more information about Near Wall Treatments in the k-epsilon model refer to the Fluent User's Guide.

iii. Click **OK** to retain the other default settings, enable the model, and close the **Viscous Model** dialog box.

Note that the **Viscous...** label in the ribbon is displayed in blue to indicate that the Viscous model is enabled.

3. Define a heat source cell zone condition for the evaporator volume.

#### $\blacksquare$ Setting Up Physics $\rightarrow$ Zones $\rightarrow$ Cell Zones

#### Note

All cell zones defined in your simulation are listed in the **Cell Zone Conditions** task page and under the **Setup/Cell Zone Conditions** tree branch.

a. In the **Cell Zone Conditions** task page, under the **Zone** list, select **fluid-evaporator** and click **Edit...** to open the **Fluid** dialog box.

E Fluid							<b>×</b>
Zone Name							
fluid-evaporator							
Material Name air	▼ Edit						
E Frame Motion E 3D Fan Zone	e 🗹 Source Terms						
🗌 Mesh Motion 🛛 Laminar Zon	e 📄 Fixed Values						
Porous Zone						_	
Reference Frame Mesh Mot	ion Porous Zone	3D Fan Zone	Embedded LES	Reaction	Source Terms	Fixed Values	Multiphase
Mass X Momentum	0 sources 0 sources	Edit					
Y Momentum	0 sources	Edit					
Z Momentum	0 sources	Edit					
Turbulent Kinetic Energy	0 sources	Edit					
Turbulent Dissipation Rate	0 sources	Edit					
Energy	1 source	Edit					
		OK	Cancel Help				

- b. In the **Fluid** dialog box, enable **Source Terms**.
- c. In the **Source Terms** tab, click the **Edit...** button next to **Energy**.

Energy sources	<b>—</b>
	Number of Energy sources 1
1. (w/m3) -787401.6	⊂onstant ▼
ОК Са	ncel Help

- d. In the Energy sources dialog box, change the Number of Energy sources to 1.
- e. For the new energy source, select **constant** from the drop-down list, and enter -787401.6 W/m<sup>3</sup> based on the evaporator load (200 W) divided by the evaporator volume (0.000254 m<sup>3</sup>) that was computed earlier.
- f. Click **OK** to close the **Energy Source** dialog box.
- g. Click **OK** to close the **Fluid** dialog box.

## 2.4.4. Defining Input Parameters in ANSYS Fluent

You have now started setting up the CFD analysis using ANSYS Fluent. In this step, you will define boundary conditions and input parameters for the velocity inlet.

- 1. Define an input parameter called in\_velocity for the velocity at the inlet boundary.
  - a. In the Setting Up Physics tab, click Boundaries (Zones group).

Setting Up Physics  $\rightarrow$  Zones  $\rightarrow$  Boundaries

This opens the Boundary Conditions task page.

#### Note

All boundaries defined in the case are also displayed under the **Setup/Boundary Conditions** tree branch.

- b. In the Boundary Conditions task page, click the Toggle Tree View button (in the upper right corner), and under the Group By category, select Zone Type. This displays boundary zones grouped by zone type.
- c. Under the Inlet zone type, double-click inlet-air.

💶 Velocity Inle	t						×
Zone Name							
inlet-air							
Momentum	Thermal	Radiation	Species	DPM	Multiphase	Potential	UDS
Velocit	y Specification I	Method Ma	gnitude, No	rmal to B	oundary		•
	Reference	Frame Ab	solute				•
	Velocity Magr	nitude (m/s	0.5		in_ve	locity	•
Supersonic/In	itial Gauge Press	sure (pascal	) 0		const	tant	-
	- Turbulence						
	Specification Method Intensity and Hydraulic Diameter					•	
	Turbulent Intensity (%) 5					P	
Hydraulic Diameter (m) 0.061							
	OK Cancel Help						

d. In the Velocity Inlet dialog box, from the Velocity Magnitude drop-down list, select New Input Parameter....

This displays the Input Parameter Properties dialog box.

💶 Input Parameter Properties 🛛 🗾	
Name	
in_velocity	-
Current Value (m/s)	
0.5	
Used In:	
	1
OK Cancel Help	

- e. Enter in\_velocity for the Name, and enter 0.5 m/s for the Current Value.
- f. Click OK to close the Input Parameter Properties dialog box.
- g. Under the **Turbulence** group box, from the **Specification Method** drop-down list, select **Intensity** and Hydraulic Diameter.
- h. Retain the value of 5 % for **Turbulent Intensity**.

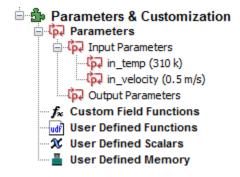
- i. Enter 0.061 for **Hydraulic Diameter (m)**.
- 2. Define an input parameter called in\_temp for the temperature at the inlet boundary.

Velocity Inlet							×
Zone Name							
inlet-air							
Momentum	Thermal	Radiation	Species	DPM	Multiphase	Potential	UDS
Temperature (k)	310		in_tem	р	-		
OK Cancel Help							

- a. In the **Thermal** tab of the **Velocity Inlet** dialog box, select **New Input Parameter...** from the **Temperature** drop-down list.
- b. Enter in\_temp for the Name and enter 310 K for the Current Value in the Input Parameter Properties dialog box.
- c. Click OK to close the Input Parameter Properties dialog box.
- d. Click **OK** to close the **Velocity Inlet** dialog box.
- 3. Review all of the input parameters that you have defined in ANSYS Fluent under the **Parameters & Cus-tomization/Parameters** tree branch.

► Parameters & Customization → Parameters → Input Parameters

Figure 2.13: The Input Parameters Sub-Branch in ANSYS Fluent



These parameters are passed to ANSYS Fluent component system in ANSYS Workbench and are available for editing in ANSYS Workbench (see Figure 2.14: The Parameters View in ANSYS Workbench (p. 95)).

Outline	of All Parameters			₹ ₽
	А	В	с	D
1	ID	Parameter Name	Value	Unit
2	Input Parameters			
3	🖃 🔯 Fluid Flow (FLUENT) (A1)			
4	ြီး P1	hcpos	25	
5	ί <mark>ρ</mark> Ρ2	ftpos	25	
6	ြို့ P3	wsfpos	90	
7	ί <mark>ρ</mark> Ρ7	in_velocity	0.5	m s^-1 💌
8	<mark>ф</mark> Р9	in_temp	310	к 💌
9	С <mark>р</mark> Р4	input_hcpos	15	
10	<mark>Ґр</mark> Р5	input_ftpos	25	
11	🛱 Рб	input_wsfpos	90	
*	🗘 New input parameter	New name	New expression	
13	Output Parameters			
*	New output parameter		New expression	
15	Charts			

Figure 2.14: The Parameters View in ANSYS Workber	ch
---	----

- 4. Set the turbulence parameters for backflow at the front outlets and foot outlets.
  - a. In the **Boundary Conditions** task page, type **outlet** in the **Zone** filter text entry field. Note that as you type, the names of the boundary zones beginning with the characters you entered appear in the boundary condition zone list.

#### Note

- The search string can include wildcards. For example, entering **\*let**\* will display all zone names containing let, such as inlet and outlet.
- To display all zones again, click the red **X** icon in the **Zone** filter.
- b. Double-click **outlet-front-mid**.

	Pressure Outle	t						<b>—</b> ×
Z	one Name							
0	utlet-front-mid							
Γ	Momentum	Thermal	Radiation	Species	DPM	Multiphase	Potential	UDS
	Backflow Reference Frame Absolute							
	Gauge Pressure (pascal) 0 constant							
E	Backflow Direction Specification Method Normal to Boundary							
	Backflow Pressure Specification Total Pressure							
	Radial Equilibrium Pressure Distribution							
	Average Pres	sure Specif	ication					
	Target Mass F	Flow Rate						
	Turbulence							
		Specificat	ion Method []	ntensity and	d Hydraul	ic Diameter		•
	Backflow Turbulent Intensity (%) 5							
	Backflow Hydraulic Diameter (m) 0.044							
L								
	OK Cancel Help							

- c. In the **Pressure Outlet** dialog box, under the **Turbulence** group box, select **Intensity and Hydraulic Diameter** from the **Specification Method** drop-down list.
- d. Retain the value of 5 for **Backflow Turbulent Intensity (%)**.
- e. Enter 0.044 for Backflow Hydraulic Diameter (m).

These values will only be used if reversed flow occurs at the outlets. It is a good idea to set reasonable values to prevent adverse convergence behavior if backflow occurs during the calculation.

- f. Click OK to close the Pressure Outlet dialog box.
- g. Copy the boundary conditions from **outlet-front-mid** to the other front outlet.

**F** Setup  $\rightarrow$  Boundary Conditions  $\rightarrow$  outlet-front-mid  $\stackrel{\textcircled{U}}{\rightarrow}$  Copy...

	Copy Conditions		×
Fro	m Boundary Zone Filter Text	F.	To Boundary Zones Filter Text 🗾 🔂 🗮 🐺
	interior-fluid-central-unit-fluid-heat-excha	*	outlet-foot-left
	interior-fluid-evaporator		outlet-front-side-left
	interior-fluid-heat-exchanger	=	outlet-windshield
4	Outlet		
	outlet-foot-left		
	outlet-front-mid		
	outlet-front-side-left		
	outlet-windshield	+	
	· ·		
	Сор	y	Close Help

- i. Confirm that outlet-front-mid is selected in the From Boundary Zone selection list.
- ii. Select outlet-front-side-left in the To Boundary Zones selection list.
- iii. Click **Copy** to copy the boundary conditions.

Fluent will display a dialog box asking you to confirm that you want to copy the boundary conditions.

- iv. Click **OK** to confirm.
- v. Close the Copy Conditions dialog box.
- h. In a similar manner, set the backflow turbulence conditions for **outlet-foot-left** using the values in the following table:

Parameter	Value
Specification Method	Intensity and Hydraulic Diameter
Backflow Turbulent Intensity (%)	5
Backflow Hydraulic Diameter (m)	0.052

## 2.4.5. Solving

In the steps that follow, you will set up and run the calculation using the **Solving** ribbon tab.

#### Note

You can also use the task pages listed under the **Solution** branch in the tree to perform solution-related activities.

1. Set the Solution Methods.





This will open the **Solution Methods** task page.

Solution Methods	
Pressure-Velocity Coupling	
Scheme	
Coupled 👻	
Spatial Discretization	
Gradient	1
Least Squares Cell Based 🔹	
Pressure	
PRESTO!	
Momentum	Ξ
First Order Upwind 🔹	
Turbulent Kinetic Energy	
First Order Upwind 🔹	
Turbulent Dissipation Rate	
First Order Upwind 🔹	
Transient Formulation	
Non-Iterative Time Advancement	
Frozen Flux Formulation	
Pseudo Transient	
Warped-Face Gradient Correction	
High Order Term Relaxation Options	
Default	
Help	

a. From the **Scheme** drop-down list, select **Coupled**.

The pressure-based coupled solver is the recommended choice for general fluid flow simulations.

b. In the **Spatial Discretization** group box, configure the following settings:

Setting	Value
Pressure	PRESTO!
Momentum	First Order Upwind
Energy	First Order Upwind

This tutorial is primarily intended to demonstrate the use of parameterization and design points when running Fluent from Workbench. Therefore, you will run a simplified analysis using first order discretization, which will yield faster convergence. These settings were chosen to speed up solution time for this tutorial. Usually, for cases like this, we would recommend higher order discretization settings to be set for all flow equations to ensure improved results accuracy.

2. Initialize the flow field using the **Initialization** group of the **Solving** ribbon tab.

	5		
	Initializ	ation	
Method		Patch	
O Hybrid	More Settings	Reset Statistics	
Standard	Options		t = 0
		Reset DPM	Initialize

- a. Retain the default selection of Hybrid Initialization.
- b. Click the **Initialize** button.

Solving  $\rightarrow$  Initialization

3. Run the simulation in ANSYS Fluent from the Run Calculation group of the Solving tab.

### Solving → Run Calculation

	Run Calculation				
Update Dynamic Mesh	No. of Itorations 1000	× <u>/</u> ,			
Input Summary	No. of Iterations 1000 🚖				
Advanced	Check Case	Calculate			

- a. For **Number of Iterations**, enter 1000.
- b. Click the **Calculate** button.

The solution converges within approximately 55 iterations.

Throughout the calculation, Fluent displays a warning in the console regarding reversed flow at the outlets. This behavior is expected in this case since air is redirected to the outlets, creating small regions of recirculation.

#### Note

The warning message can be switched off by setting the <code>solve/set/flow-warnings</code> text user interface (TUI) command to no in the console.

4. Close Fluent.

File → Close Fluent

5. Save the project in ANSYS Workbench.

#### $\mathbf{File} \rightarrow \mathbf{Save}$

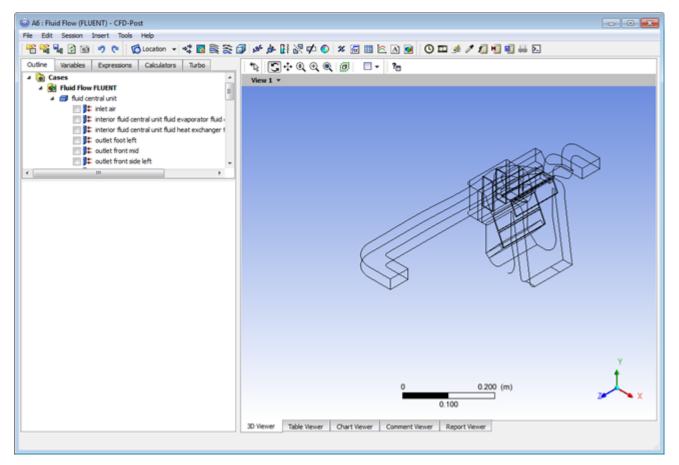
## 2.4.6. Postprocessing and Setting the Output Parameters in ANSYS CFD-Post

In this step, you will visualize the results of your CFD simulation using ANSYS CFD-Post. You will plot vectors that are colored by pressure, velocity, and temperature, on a plane within the geometry. In addition, you will create output parameters within ANSYS CFD-Post for later use in ANSYS Workbench.

In the ANSYS Workbench **Project Schematic**, double-click the **Results** cell in the ANSYS Fluent fluid flow analysis system to start CFD-Post. You can also right-click the **Results** cell to display the context menu where you can select the **Edit...** option.

The CFD-Post application appears with the automotive HVAC geometry already loaded and displayed in outline mode. Note that ANSYS Fluent results (that is, the case and data files) are also automatically loaded into CFD-Post.





1. Edit some basic settings in CFD-Post (for example, changing the background color to white).

#### $\textbf{Edit} \rightarrow \textbf{Options...}$

a. In the **Options** dialog box, select **Viewer** under **CFD-Post** in the tree view.

🛛 🔽 Object Hig	hlighting	
Туре	Surface Mesh	
Pre-genera	ate region highlight	
Background		
Mode	Color	
Color Type	Solid	
Color		]
ANSYS Logo	White	
Text Color		
Edge Color		
Axis Visibility 🛛 🖉 Ruler Visibilit		
Stereo		
Mode	Normal	
Stereo Effect	Weaker	
For stereo viev	reo <b>Mode</b> setting only takes effect the next time you run the application. wing to work, you need to turn on 'Stereo' in your graphic card, have a display that supports stereo, and ensure is set to Perspective mode (Right-click in viewer > Projection > Perspective).	e

- b. Under the Background group, from the **Color Type** drop-down list, select **Solid**.
- c. Click the **Color** sample bar to cycle through common color swatches until it displays white.

Tip

You can also click the ellipsis icon which you can choose an arbitrary color.

- d. Click **OK** to set the white background color for the display and close the **Options** dialog box.
- 2. Adjust the color-map legend to show the numbers in floating format.
  - a. In the **Outline** tree view, double-click **Default Legend View 1** to display the **Details** view for the default legend to be used for your plots.
  - b. In the **Definition** tab of the **Details** view, from the **Title Mode** drop-down list, select **Variable**.

#### Details of Default Legend View 1

Definition A	ppearance
Title Mode	Variable 👻
<ul><li>Show Lege</li><li>Vertical</li></ul>	<ul> <li>Horizontal</li> </ul>
Location	
X Justification	Left 🔹
Y Justification	Тор 👻
Position	0.02 0.15

c. In the Appearance tab, set the Precision to 2 and Fixed.

Definition	Appearance
Sizing Para	meters
Size	0.6
Aspect	0.07
- Text Param	eters
Precision	2 Fixed -
Value Ticks	5
Font	Sans Serif 👻
Color Mode	Default
Colour	
Text Rotatio	n O
Text Height	0.024

Details of Default Legend View 1

- d. Click **Apply** to set the display.
- 3. Plot vectors colored by pressure.
  - a. From the main menu, select **Insert**  $\rightarrow$  **Vector** or click  $\Rightarrow$  in the ANSYS CFD-Post toolbar.

This displays the **Insert Vector** dialog box.

- b. Keep the default name of **Vector 1** by clicking **OK**.
- c. In the **Details** view for **Vector 1**, under the **Geometry** tab, configure the following settings.

Details of <b>Vecto</b>	r 1				
Geometry	Color	Symbol	Render	View	
Domains	All Dor	mains			•
Definition					
Locations	symm	netry central	unit		<b>•</b>
Sampling	Equa	lly Spaced			•
# of Points	1000	0			<b>*</b>
Variable	Veloc	ity			•
Boundary Data	Э	🔘 Hybrid		Ocn:	servative
Projection	Tang	ential			<b>•</b>

- i. Ensure All Domains is selected from the Domains drop-down list.
- ii. From the Locations drop-down list, select symmetry central unit.
- iii. From the **Sampling** drop-down list, select **Equally Spaced**.
- iv. Set the **# of Points** to 10000.
- v. From the **Projection** drop-down list, select **Tangential**.
- d. In the **Color** tab, configure the following settings.

Geometry	Color	Symbol	Render	View	
Mode	Variable	e			-
Variable	Press	ure			•
Range	Globa				•
Min				u	nknown
Max				u	nknown
Boundary Dat	ta	🔘 Hybrid		One Construction	servative
Color Scale	Linear	,			•
Color Map	Defau	ılt (Rainbow	)		- 8
Undef. Color					

Details of **Vector 1** 

i. From the **Mode** drop-down list, select **Variable**.

- ii. From the Variable drop-down list, select Pressure.
- e. In the **Symbol** tab, configure the following settings.

#### Details of **Vector 1**

Geometry	Color	Symbol	Render	View	
Symbol	Line Ar	row			•
Symbol Size	0.05				
🔽 Normalize	9 Symbols				

- i. Set the **Symbol Size** to 0.05.
- ii. Enable Normalize Symbols.
- f. Click Apply.

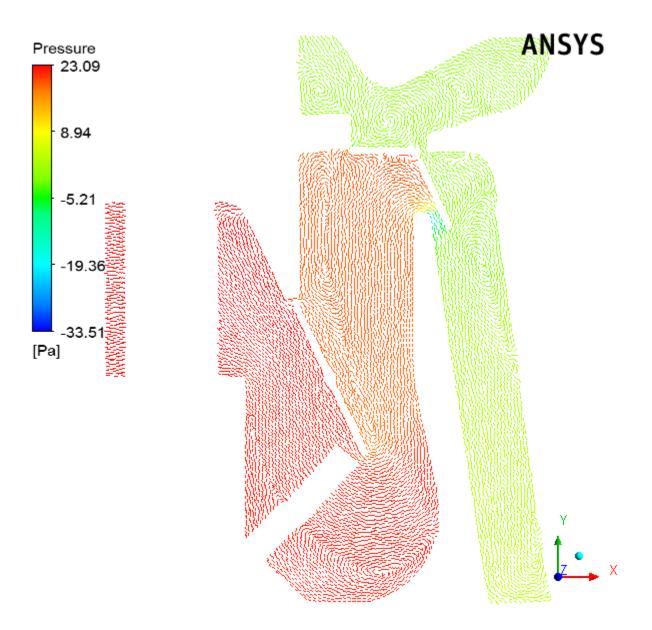
Vector 1 appears under User Locations and Plots in the Outline tree view.

In the graphics display window, note that **symmetry-central-unit** shows the vectors colored by pressure. Use the controls in CFD-Post to rotate the geometry (for example, clicking the dark blue axis in the axis triad of the graphics window). Zoom into the view as shown in Figure 2.16: Vectors Colored by Pressure (p. 105).

#### Note

To better visualize the vector display, you can deselect the **Wireframe** view option under **User Locations and Plots** in the **Outline** tree view.

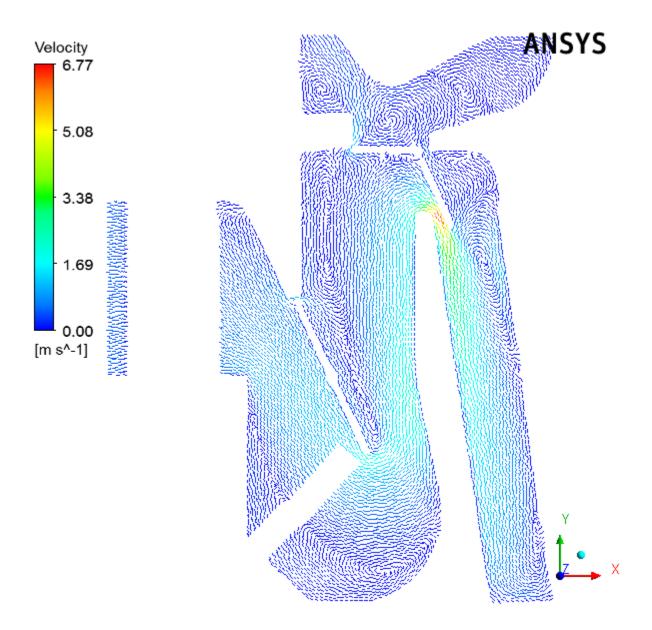




- 4. Plot vectors colored by velocity.
  - a. In the **Details** view for **Vector 1**, under the **Color** tab, configure the following settings.
    - i. Select **Velocity** from the **Variable** drop-down list.
    - ii. Click Apply.

The velocity vector plot appears on the **symmetry-central-unit** symmetry plane.

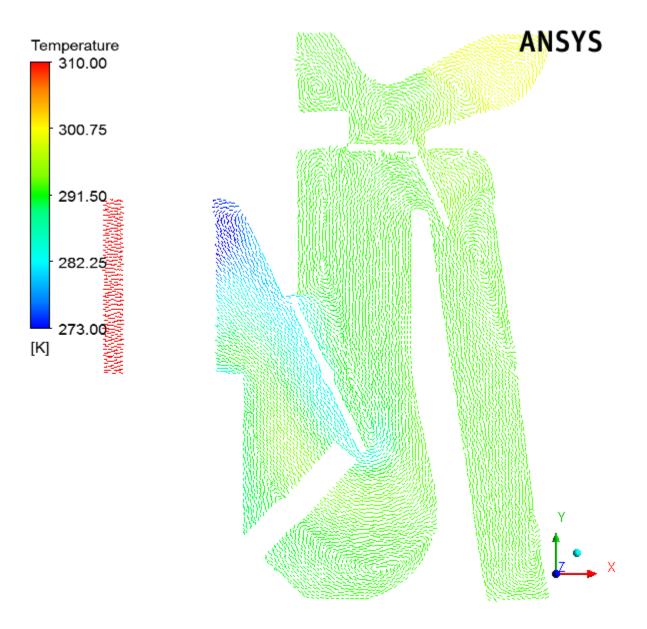




- 5. Plot vectors colored by temperature.
  - a. In the **Details** view for **Vector 1**, under the **Color** tab, configure the following settings.
    - i. Select Temperature from the Variable drop-down list.
    - ii. Select User Specified from the Range drop-down list.
    - iii. Enter 273 K for the **Min** temperature value.
    - iv. Enter 310 K for the Max temperature value.
    - v. Click Apply.

The user-specified range is selected much narrower than the Global and Local ranges in order to better show the variation.





Note the orientation of the various valves and how they impact the flow field. Later in this tutorial, you will change these valve angles to see how the flow field changes.

6. Create two surface groups.

Surface groups are collections of surface locations in CFD-Post. In this tutorial, two surface groups are created in CFD-Post that will represent all of the outlets and all of the front outlets. Once created, specific commands (or expressions) will be applied to these groups in order to calculate a particular numerical value at that surface.

- a. Create a surface group consisting of all outlets.
  - i. With the **Outline** tree view open in the CFD-Post tree view, open the **Insert Surface Group** dialog box.

#### Insert $\rightarrow$ Location $\rightarrow$ Surface Group

💿 Ins	ert Surface Group 📑 🞫
Name	alloutlets
	DK Cancel

- ii. Enter alloutlets for the Name of the surface group, and click OK to close the Insert Surface Group dialog box.
- iii. In the **Details** view for the alloutlets surface group, in the **Geometry** tab, click the ellipsis icon

mext to **Locations** to display the **Location Selector** dialog box.

Location Selector
▲ Fluid Flow FLUENT
Ĵ‡ inlet air
Ĵ‡ interior fluid central unit fluid evaporator fluid central unit
Ĵ‡ interior fluid central unit fluid evaporator fluid evaporator
Ĵ‡ interior fluid central unit fluid heat exchanger fluid central unit
🕽 🗱 interior fluid central unit fluid heat exchanger fluid heat exchanger
Ĵ‡ outlet foot left
Ĵ‡ outlet front mid
Ĵ‡ outlet front side left
Ĵ‡ outlet windshield
🕽 🗱 symmetry central unit
Ĵ‡ symmetry evaporator
🕽 🗱 symmetry heat exchanger
)‡ wall central unit
)‡ wall evaporator
Ĵ‡ wall ftpos
)‡ wall hcpos
🕽 😂 wall heat exchanger
)‡ wall inlet flow diverter
Ĵ‡ wall wsfpos
OK <u>C</u> ancel

- iv. Select all of the outlet surfaces (outlet foot left, outlet front mid, outlet front side left, and outlet windshield) in the Location Selector dialog box (hold Ctrl for multiple selection) and click OK.
- v. Click **Apply** in the **Details** view for the new surface group.

alloutlets appears under User Locations and Plots in the Outline tree view.

b. Create a surface group for the front outlets.

Perform the same steps as described above to create a surface group called frontoutlets with locations for the front outlets (outlet front mid and outlet front side left).

7. Create expressions in CFD-Post and mark them as ANSYS Workbench output parameters.

In this tutorial, programmatic commands or expressions are written to obtain numerical values for the mass flow rate from all outlets, as well as at the front outlets, windshield, and foot outlets. The surface groups you defined earlier are used to write the expressions.

- a. Create an expression for the mass flow from all outlets.
  - i. With the **Expressions** tab open in the CFD-Post tree view, open the **Insert Expression** dialog box.

#### **Insert** → **Expression**

. . . . . .

ii. Enter floutfront for the Name of the expression and click OK to close the Insert Expression dialog box.

Details of flou	ittront		
Definition	Plot	Evaluate	
-(massFlow	v()@fror	toutlets)*2	2
Value		-0.000264	112 [kg s^-1]
Apply			Reset

iii. In the **Details** view for the new expression, enter the following in the **Definition** tab.

-(massFlow()@frontoutlets)\*2

The sign convention for **massFlow()** is such that a positive value represents flow into the domain and a negative value represents flow out of the domain. Since you are defining an expression for outflow from the ducts, you use the negative of the **massFlow()** result in the definition of the expression.

iv. Click **Apply** to obtain a **Value** for the expression.

Note the new addition in the list of expressions in the **Expressions** tab in CFD-Post.

In this case, there is a small net backflow into the front ducts.

- v. Right-click the new expression and select **Use as Workbench Output Parameter** from the context menu. A small "P" with a right-pointing arrow appears on the expression's icon.
- b. Create an expression for the mass flow from the wind shield.
  - i. Perform the same steps as described above to create an expression called floutwindshield with the following definition:

-(massFlow()@outlet windshield)\*2

- ii. Right-click the new expression and select **Use as Workbench Output Parameter** from the context menu.
- c. Create an expression for the mass flow from the foot outlets.
  - i. Perform the same steps as described above to create an expression called floutfoot with the following definition:

```
-(massFlow()@outlet foot left)*2
```

- ii. Right-click the new expression and select **Use as Workbench Output Parameter** from the context menu.
- d. Create an expression for the mass weighted average outlet temperature.
  - i. Perform the same steps as described above to create an expression called outlettemp with the following definition:

massFlowAveAbs(Temperature)@alloutlets

- ii. Right-click the new expression and select **Use as Workbench Output Parameter** from the context menu.
- 8. Close ANSYS CFD-Post.

In the main menu, select **File**  $\rightarrow$  **Close CFD-POST** to return to ANSYS Workbench.

- 9. In the **Outline of All Parameters** view of the **Parameter Set** tab (double-click **Parameter Set**), review the newly-added output parameters that you specified in ANSYS CFD-Post and when finished, click the **Project** tab to return to the **Project Schematic**.
- 10. If any of the cells in the analysis system require attention, update the project by clicking the **Update Project** button in the ANSYS Workbench toolbar.
- 11. Optionally, review the list of files generated by ANSYS Workbench. If the **Files** view is not open, select **View** → **Files** from the main menu.

You will notice additional files associated with the latest solution as well as those generated by CFD-Post.

File View Tools Units Extensions	Jobs H	Help			
🗋 🚰 🛃 🦳 Project 🙀 Pa	rameter Sel	×			
🚮 Import 🗟 Refresh Pr	oject 🍠 U	Ipdate Project 🧚 Update All Design Points	ACT Start	Page	
oobox 🔻 🦊	X Project	t Schematic			
E Analysis Systems					
G Fluid Flow (Fluent)	-				
IC Engine (Fluent)		▼ A			
🚽 Throughflow		1 🖸 Fluid Flow (FLUENT)			
🪽 Throughflow (BladeGen)		2 🝈 Geometry 🗸			
Component Systems		3 🍘 Mesh 🗸 .			
Design Exploration		4 🍓 Setup 🗸			
External Connection Systems					
	- 1	5 🗑 Solution			
		6 💓 Results 🗸 🖌			
		> 7 🛱 Parameters			
		Fluid Flow (FLUENT)			
	<del>م</del> ە	Parameter Set			
	්ත Files	Parameter Set			
		Parameter Set	в	с	D
			-	C Size 💌	-
	Files	A	-	-	Туре
	Files	A Name	-	Size 🔻	Туре
	Files	A Name •	Ce 💌	Size	Type Workbench Project F
	Files 1 2 3	A Name fluent-workbench-param.wbpj Geom.agdb	Ce •	Size 235 KB 2 MB	Type Workbench Project F Geometry File
	Files 1 2 3 4	A Name  fluent-workbendh-param.wbpj  Geom.agdb  FFF.agdb	Ce         ▼           A2         A2	Size  235 KB 2 MB 2 MB 2 MB	Type Workbench Project F Geometry File Geometry File
	Files 1 2 3 4 5	A Name  fluent-workbench-param.wbpj  Geom.agdb  FFF.agdb  FFF.mshdb	Ce	Size ▼ 235 KB 2 MB 2 MB 28 MB	Type Workbench Project F Geometry File Geometry File .mshdb
	Files 1 2 3 4 5 6	A Name  fluent-workbench-param.wbpj  Geom.agdb  FFF.agdb  FFF.mshdb  FFF.msh	Ce	Size	Type Workbench Project F Geometry File .mshdb Filuent Mesh File
	Files 1 2 3 4 5 6 7	A Name  Name  fuent-workbench-param.wbpj  Geom.agdb  FFF.agdb  FFF.mshdb  FFF.msh  FFF.msh  FFF.set  FFF.1.cas.gz	Ce A2 A2 A3 A3,A4 A4	Size	Type Workbench Project F Geometry File .mshdb Fiuent Mesh File FLUENT Model File
	Files 1 2 3 4 5 6 7 8	A Name  Name  fluent-workbench-param.wbpj  Geom.agdb  FFF.agdb  FFF.mshdb  FFF.msh  FFF.set  FFF.set  FFF-1.cas.gz	Ce	Size            235 KB         2 MB           2 MB         2 MB           28 MB         73 MB           217 KB         30 MB	Type Workbench Project F Geometry File Geometry File .mshdb Fluent Mesh File FLUENT Model File FLUENT Case File
	Files 1 2 3 4 5 6 7 8 9 10	A Name Name Kiewerkbench-param.wbpj Geom.agdb FFF.agdb FFF.mshdb FFF.msh FFF.set FFF.1-cas.gz FFF-1-cos5s.dat.gz FFF-1-00055.dat.gz FFF-1-00055.dat.gz	Ce	Size            235 KB         2 MB           2 MB         2 MB           2 MB         2 MB           217 KB         30 MB           36 MB         36 MB	Type Workbench Project F Geometry File Geometry File Jmshdb Filuent Mesh File FLUENT Model File FLUENT Case File FLUENT Data File CFD-Post State File
	Files 1 2 3 4 5 6 7 8 9	A Name Name Kinent-workbench-param.wbpj Geom.agdb FFF.agdb FFF.mshdb FFF.msh FFF.set FFF.set FFF-1.cas.gz FFF-1-00055.dat.gz FFF-1-00055.dat.gz FHId Flow FLUENT.cst	Ce	Size            235 KB         2 MB           2 MB         2 MB           28 MB         73 MB           217 KB         30 MB           36 MB         36 MB	Type Workbench Project F Geometry File Geometry File .mshdb Fluent Mesh File FLUENT Model File FLUENT Case File FLUENT Data File

#### Figure 2.19: The Updated Project Loaded into ANSYS Workbench Displaying the Files View

12. Save the project in ANSYS Workbench.

In the main menu, select **File**  $\rightarrow$  **Save** 

#### Note

You can also select the Save Project option from the CFD-Post File ribbon tab.

## 2.4.7. Creating Additional Design Points in ANSYS Workbench

Parameters and design points are tools that allow you to analyze and explore a project by giving you the ability to run optimization and what-if scenarios. Design points are based on sets of parameter values. When you define input and output parameters in your ANSYS Workbench project, you are essentially working with a design point. To perform optimization and what-if scenarios, you create multiple

design points based on your original project. In this step, you will create additional design points for your project where you will be able to perform a comparison of your results by manipulating input parameters (such as the angles of the various valves within the automotive HVAC geometry). ANSYS Workbench provides a Table of Design Points to make creating and manipulating design points more convenient.

- 1. Open the Table of Design Points.
  - a. In the Project Schematic, double-click the **Parameter Set** bus bar to open the Table of Design Points view. If the table is not visible, select **Table** from the **View** menu in ANSYS Workbench.

#### View → Table

The table of design points initially contains the current project as a design point (DP0), along with its corresponding input and output parameter values.

Figure 2.20: Table of Design Points (with DP0)

Table of Design Points										
	A	в	С	D	E	F	G	н	I	
1	Name 💌	P1 - hcpos 💌	P2 - ftpos 💌	P3 - wsfpos 💌	P4 - input_hcpos 💌	P5 - input_ftpos 💌	P6 - input_wsfpos 💌	P7 - in_velocity 💌	P8 - in_temp 💌	
2	Units							m s^-1 💌	к 💌	
3	DP 0 (Current)	25	25	90	15	25	90	0.5	310	

From this table, you can create new design points (or duplicate existing design points) and edit them (by varying one or more input parameters) to create separate analyses for future comparison of data.

- 2. Create a design point (DP1) by duplicating the current design point (DP0).
  - a. Right-click the **Current** design point and select **Duplicate Design Point** from the context menu.

The cells autofill with the values from the **Current** row.

b. Scroll over to the far right to expose the **Retain** column in the table of design points, and ensure the check box in the row for the duplicated design point **DP 1** (cell N4) is selected.

This allows the data from this new design point to be saved before it is exported for future analysis.

- 3. Create another design point (DP2) by duplicating the DP1 design point.
  - a. Right-click the **DP1** design point and select **Duplicate Design Point** from the context menu.

Since this is a duplicate of DP1, this design point will also have its data retained.

4. Edit values for the input parameters for DP1 and DP2.

For DP1 and DP2, edit the values for your input parameters within the Table of Design Points as follows:

	input_hcpos	input_ftpos	input_wsfpos	in_velocity	in_temp
DP1	45	45	45	0.6	300
DP2	90	60	15	0.7	290

Table of Design Points										
	A	в	с	D	E	F	G	н	I	
1	Name 💌	P1 - hcpos 💌	P2 - ftpos 💌	P3 - wsfpos 💌	P4 - input_hcpos 💌	P5 - input_ftpos 💌	P6 - input_wsfpos 💌	P7 - in_velocity 💌	P8 - in_temp 💌	
2	Units							m s^-1	к 💌	
3	DP 0 (Current)	25	25	90	15	25	90	0.5	310	
4	DP 1	45	45	45	45	45	45	0.6	300	
5	DP 2	90	60	15	90	60	15	0.7	290	

Figure 2.21: Table of Design Points (with DP0, DP1, and DP2 Defined)

For demonstration purposes of this tutorial, in each design point, you are slightly changing the angles of each of the valves, and increasing the inlet velocity and the inlet temperature. Later, you will see how the results in each case vary.

5. Update all of your design points.

Click the **Update all Design Points** button in the ANSYS Workbench toolbar. Alternatively, you can also select one or more design points, right-click, and select **Update Selected Design Points** from the context menu. Click **OK** to acknowledge the information message notifying you that some open editors may close during the update process. By updating the design points, ANSYS Workbench takes the new values of the input parameters for each design point and updates the components of the associated system (for example, the geometry, mesh, settings, solution, and results), as well as any output parameters that have been defined.

#### Note

It may take significant time and/or computing resources to re-run the simulations for each design point.

6. Export the design points to separate projects.

This will allow you to work with calculated data for each design point.

- a. Select the three design points, DP0, DP1, and DP2 (hold Shift for multiple selection).
- b. Right-click the selected design points and select **Export Selected Design Points**.

Note the addition of three more ANSYS Workbench project files (and their corresponding folders) in your current working directory (fluent-workbench-param\_dp0.wbpj, fluent-workbench-param\_dp1.wbpj and fluent-workbench-param\_dp2.wbpj). You can open each of these projects up separately and examine the results of each parameterized simulation.

#### Tip

You can easily access files in your project directory directly from the **Files** view by right-clicking any cell in the corresponding row and selecting **Open Containing Folder** from the menu that opens.

7. Inspect the output parameter values in ANSYS Workbench.

Once all design points have been updated, you can use the table of design points to inspect the values of the output parameters you created in CFD-Post (for example, the mass flow parameters at the various outlets: floutfront, floutfoot, floutwindshield, and outlettemp). These, and the rest of the output parameters are listed to the far right in the table of design points.

Figure 2.22: Table of Design Points (Showing Output Parameters for DP0, DP1, and DP2)

Table of	Table of Design Points										
	A		E	F	G	н	I	J	к	L	м
1	Name 💌		p4 - input_hcpos	PS - input_ftpos	P6 - input_wsfpos	P7 - in_velocity	P8 - in_temp 💌	P9 - floutfront	P 10 - floutwindshield	P11 - floutfoot	P12 - outlettemp
2	Units					m s^-1 💌	к 💌	kg s^-1	kg s^-1	kg s^-1	к
3	DP 0 (Current)	Π	15	25	90	0.5	310	-0.00026376	0.0011285	0.01016	292.34
4	DP 1	Π	45	45	45	0.6	300	9.5785E-05	0.0071258	0.0060084	284.98
5	DP 2		90	60	15	0.7	290	0.0016708	0.0081473	0.0056169	277.13
•											

- 8. Click the **Project** tab, just above the ANSYS Workbench toolbar to return to the **Project Schematic**.
- 9. View the list of files generated by ANSYS Workbench (optional).

#### $View \rightarrow Files$

The additional files for the new design points are stored with their respective project files since you exported them.

10. Save the project in the current state in ANSYS Workbench.

In the main menu, select **File**  $\rightarrow$  **Save**.

11. Quit ANSYS Workbench.

In the main menu, select **File**  $\rightarrow$  **Exit**.

## 2.4.8. Postprocessing the New Design Points in CFD-Post

In this section, you will open the ANSYS Workbench project for each of the design points and inspect the vector plots based on the new results of the simulations.

- 1. Study the results of the first design point (DP1).
  - a. Open the ANSYS Workbench project for the first design point (DP1).

In your current working folder, double-click the fluent-workbench-param\_dpl.wbpj file to open ANSYS Workbench.

- b. Open CFD-Post by double-clicking the **Results** cell in the Project Schematic for the Fluid Flow (Fluent) analysis system.
- c. View the vector plot colored by temperature. Ensure that **Range** in the **Color** tab is set to **User Specified** and the **Min** and **Max** temperature values are set to 273 K and 310 K, respectively.

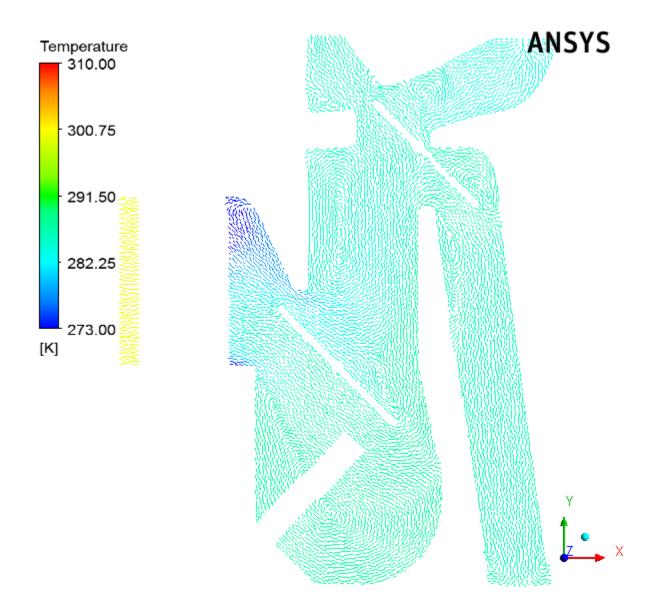
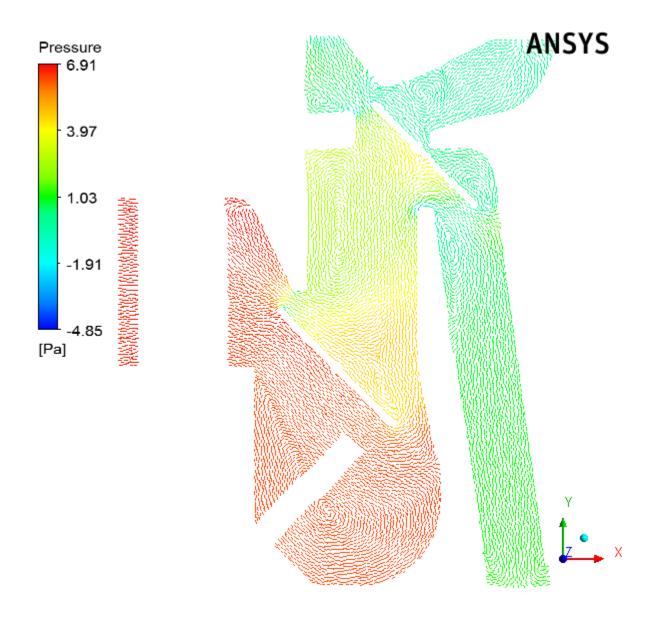


Figure 2.23: Vectors Colored by Temperature (DP1)

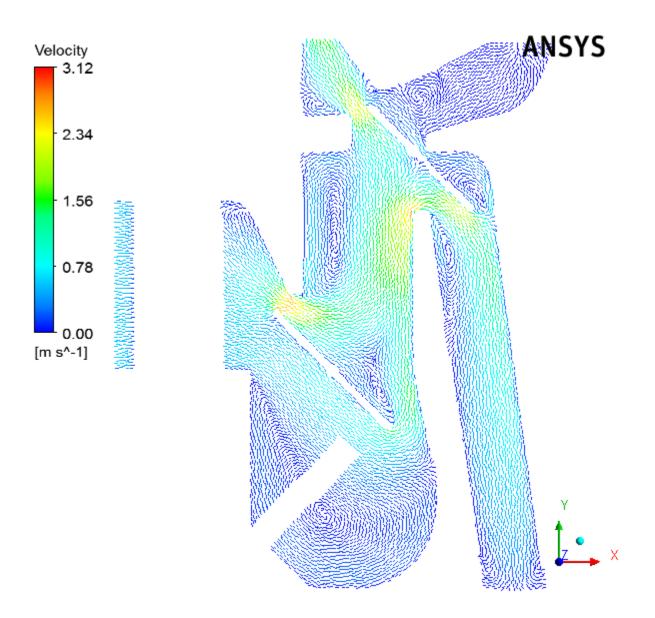
d. View the vector plot colored by pressure. Ensure that **Range** in the **Color** tab is set to **Global**.





e. View the vector plot colored by velocity. Ensure that **Range** in the **Color** tab is set to **Global**.





- f. When you are finished viewing results of the design point DP1 in ANSYS CFD-Post, select **File** → **Close CFD-Post** to quit ANSYS CFD-Post and return to the ANSYS Workbench **Project Schematic**, and then select **File** → **Exit** to exit from ANSYS Workbench.
- 2. Study the results of the second design point (DP2).
  - a. Open the ANSYS Workbench project for the second design point (DP2).

In your current working folder, double-click the fluent-workbench-param\_dp2.wbpj file to open ANSYS Workbench.

b. Open CFD-Post by double-clicking the **Results** cell in the Project Schematic for the Fluid Flow (Fluent) analysis system.

c. View the vector plot colored by temperature. Ensure that **Range** in the **Color** tab is set to **User Specified** and the **Min** and **Max** temperature values are set and the **Min** and **Max** temperature values are set to 273 K and 310 K, respectively.

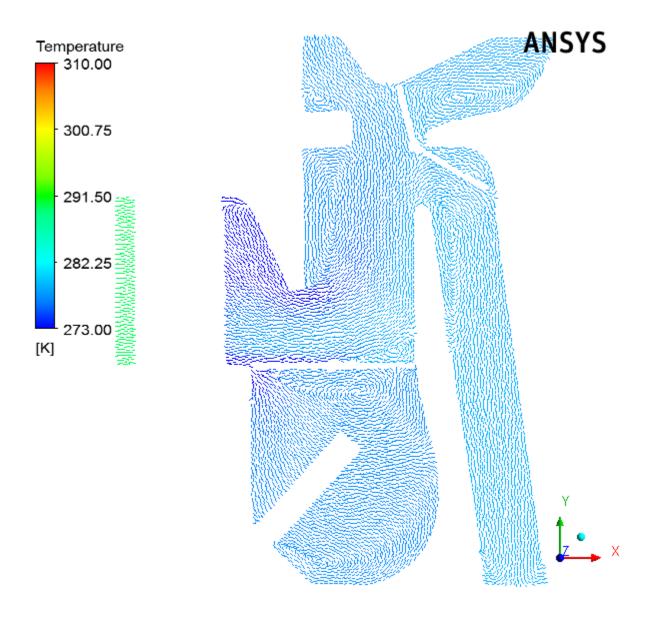


Figure 2.26: Vectors Colored by Temperature (DP2)

d. View the vector plot colored by pressure. Ensure that **Range** in the **Color** tab is set to **Global**.

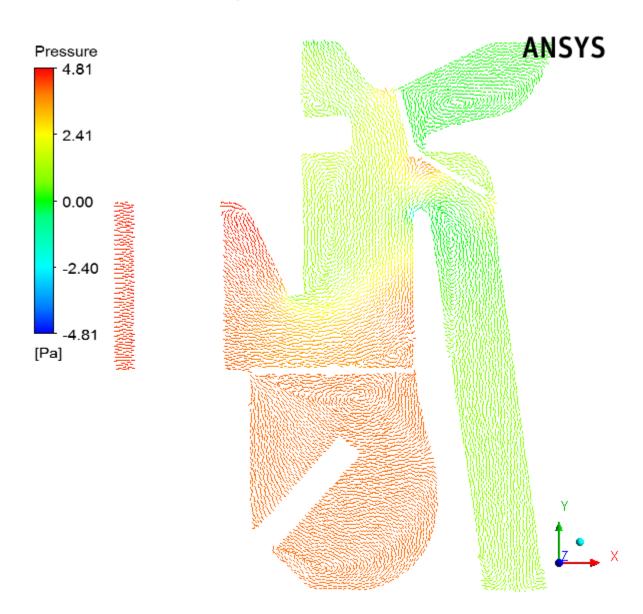
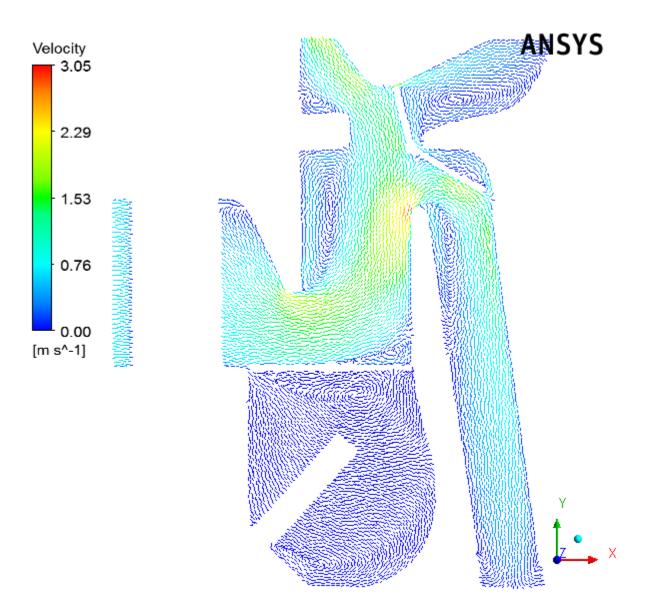


Figure 2.27: Vectors Colored by Pressure (DP2)

e. View the vector plot colored by velocity. Ensure that **Range** in the **Color** tab is set to **Global**.





3. When you are finished viewing results in ANSYS CFD-Post, select **File** → **Close CFD-Post** to quit ANSYS CFD-Post and return to the ANSYS Workbench **Project Schematic**, and then select **File** → **Exit** to exit from ANSYS Workbench.

## 2.4.9. Summary

In this tutorial, input and output parameters were created within ANSYS Workbench, ANSYS Fluent, and ANSYS CFD-Post in order to study the airflow in an automotive HVAC system. ANSYS Fluent was used to calculate the fluid flow throughout the geometry using the computational mesh, and ANSYS CFD-Post was used to analyze the results. ANSYS Workbench was used to create additional design points based on the original settings, and the corresponding simulations were run to create separate projects where parameterized analysis could be performed to study the effects of variable angles of the inlet valves, velocities, and temperatures. Also, note that simplified solution settings were used in this tutorial to speed up the solution time. For more improved solution accuracy, you would typically use denser mesh and higher order discretization for all flow equations.

# Chapter 3: Introduction to Using ANSYS Fluent: Fluid Flow and Heat Transfer in a Mixing Elbow

This tutorial is divided into the following sections:

- 3.1. Introduction
- 3.2. Prerequisites
- 3.3. Problem Description
- 3.4. Setup and Solution in Serial
- 3.5. Setup and Solution in Parallel
- 3.6. Summary

# 3.1. Introduction

This tutorial illustrates the setup and solution of a three-dimensional turbulent fluid flow and heat transfer problem in a mixing elbow. The mixing elbow configuration is encountered in piping systems in power plants and process industries. It is often important to predict the flow field and temperature field in the area of the mixing region in order to properly design the junction.

This tutorial demonstrates how to do the following:

- Launch the serial version of ANSYS Fluent.
- Read an existing mesh file into ANSYS Fluent.
- Use mixed units to define the geometry and fluid properties.
- Set material properties and boundary conditions for a turbulent forced-convection problem.
- Create a surface report definition and use it as a convergence criterion.
- Calculate a solution using the pressure-based solver.
- Visually examine the flow and temperature fields using the postprocessing tools available in ANSYS Fluent.
- Change the solver method to coupled in order to increase the convergence speed.
- Adapt the mesh based on the temperature gradient to further improve the prediction of the temperature field.
- Run the ANSYS Fluent solver in parallel.

# 3.2. Prerequisites

This tutorial assumes that you have little or no experience with ANSYS Fluent, and so each step will be explicitly described.

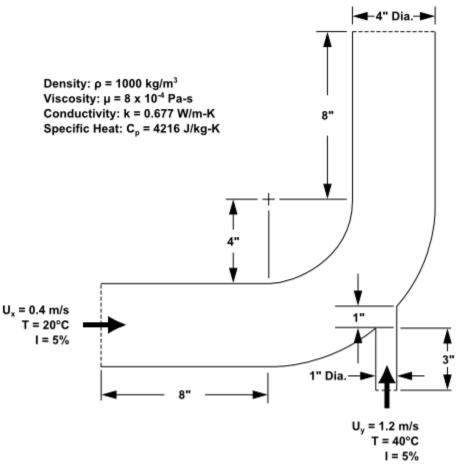
# **3.3. Problem Description**

The problem to be considered is shown schematically in Figure 3.1: Problem Specification (p. 122). A cold fluid at 20° C flows into the pipe through a large inlet, and mixes with a warmer fluid at 40° C that enters through a smaller inlet located at the elbow. The pipe dimensions are in inches and the fluid properties and boundary conditions are given in SI units. The Reynolds number for the flow at the larger inlet is 50,800, so a turbulent flow model will be required.

#### Note

Since the geometry of the mixing elbow is symmetric, only half of the elbow must be modeled in ANSYS Fluent.





## 3.4. Setup and Solution in Serial

To help you quickly identify graphical user interface items at a glance and guide you through the steps of setting up and running your simulation, the ANSYS Fluent Tutorial Guide uses several type styles and mini flow charts. See Typographical Conventions Used In This Manual (p. xvi) for detailed information.

The following sections describe the setup and solution steps for running this tutorial in serial:

- 3.4.1. Preparation
- 3.4.2. Launching ANSYS Fluent

3.4.3. Reading the Mesh
3.4.4. Setting Up Domain
3.4.5. Setting Up Physics
3.4.6. Solving
3.4.7. Displaying the Preliminary Solution
3.4.8. Using the Coupled Solver
3.4.9. Adapting the Mesh

## 3.4.1. Preparation

- 1. Set up a working folder on the computer you will be using.
- 2. Go to the ANSYS Customer Portal, https://support.ansys.com/training.

#### Note

If you do not have a login, you can request one by clicking **Customer Registration** on the log in page.

- 3. Enter the name of this tutorial into the search bar.
- 4. Narrow the results by using the filter on the left side of the page.
  - a. Click ANSYS Fluent under Product.
  - b. Click 18.0 under Version.
- 5. Select this tutorial from the list.
- 6. Click the **introduction\_R180.zip** link to download the input files.
- 7. Unzip the introduction\_R180.zip file you downloaded to your working folder. This file contains a folder, introduction, that holds the file elbow.msh that you will use in this tutorial.

#### Note

ANSYS Fluent tutorials are prepared using ANSYS Fluent on a Windows system. The screen shots and graphic images in the tutorials may be slightly different than the appearance on your system, depending on the operating system and/or graphics card.

## 3.4.2. Launching ANSYS Fluent

 From the Windows Start menu, select Start > All Programs > ANSYS 18.0 > Fluid Dynamics > Fluent 18.0 to start Fluent Launcher.

Fluent Launcher allows you to decide which version of ANSYS Fluent you will use, based on your geometry and on your processing capabilities.

E Fluent Launcher	
<b>ANSYS</b>	Fluent Launcher
Dimension ② 2D ④ 3D	Options Double Precision Meshing Mode
Display Options Display Mesh After Reading Workbench Color Scheme	Processing Options Serial Parallel
ACT Option	
吾 Show More Options	
OK Default	Cancel Help 💌

- 2. Ensure that the proper options are enabled.
  - a. Select **3D** from the **Dimension** list by clicking the radio button or the text.
  - b. Select Serial from the Processing Options list.
  - c. Ensure that the Display Mesh After Reading and Workbench Color Scheme options are enabled.

#### Note

An option is enabled when there is a check mark in the check box, and disabled when the check box is empty. To change an option from disabled to enabled (or vice versa), click the check box or the text.

d. Ensure that the **Double Precision** option is disabled.

#### Note

Fluent will retain your preferences for future sessions.

#### Extra

You can also restore the default settings by clicking the **Default** button.

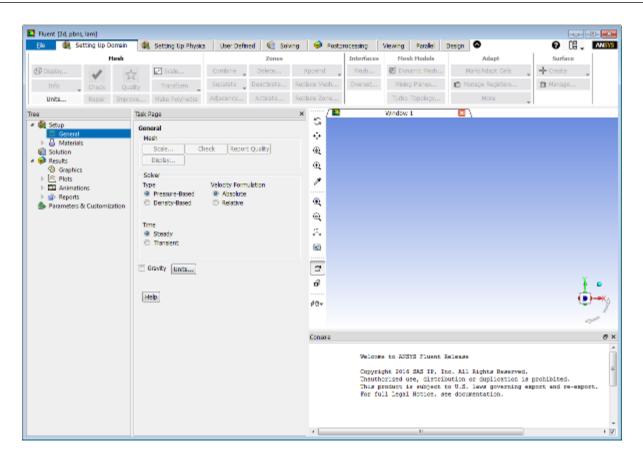
3. Set the working folder to the one created when you unzipped introduction\_R180.zip.

- a. Click the **Show More Options** button to reveal additional options.
- b. Enter the path to your working folder for **Working Directory** by double-clicking the text box and typing.

Alternatively, you can click the browse button () next to the **Working Directory** text box and browse to the directory, using the **Browse For Folder** dialog box.

Fluent Launcher	
<b>ANSYS</b>	Fluent Launcher
Dimension ② 2D ④ 3D	Options Double Precision Meshing Mode
Display Options Display Mesh After Reading Workbench Color Scheme ACT Option Load ACT	Processing Options Serial Parallel
Show Fewer Option:	eduler Environment
Version       Version       Working Directory	Pre/Post Only
C:\Tutorials\introduction Fluent Root Path	▼ 🎽
C:\Program Files\ANSYS Inc\vXX\fluent	• 📔
OK Default	Cancel Help 🔻

4. Click **OK** to launch ANSYS Fluent.



## 3.4.3. Reading the Mesh

1. Read the mesh file elbow.msh.

Click the **File** ribbon tab, then click **Read** and **Mesh...** in the menus that open in order to open the **Select File** dialog box.

File  $\rightarrow$  Read  $\rightarrow$  Mesh...

Select File	? 💌
Look in: C:\Tutorials\introduction	
Mesh File elbow.msh	ОК
Files of type: Mesh Files (*.msh* *.MSH* )	Cancel
Filter String	Filter
✓ Display Mesh After Reading	

- a. Select the mesh file by clicking **elbow.msh** in the **introduction** folder created when you unzipped the original file.
- b. Click **OK** to read the file and close the **Select File** dialog box.

As the mesh file is read by ANSYS Fluent, messages will appear in the console reporting the progress of the conversion. ANSYS Fluent will report that 13,852 hexahedral fluid cells have been read, along with a number of boundary faces with different zone identifiers.

After having completed reading mesh, ANSYS Fluent displays the mesh in the graphics window.

## Extra

You can use the mouse to probe for mesh information in the graphics window. If you click the right mouse button with the pointer on any node in the mesh, information about the associated zone will be displayed in the console, including the name of the zone.

Alternatively, you can click the probe button (📶) in the graphics toolbar and click the
left mouse button on any node. This feature is especially useful when you have several
zones of the same type and you want to distinguish between them quickly.

\_\_\_\_

For this 3D problem, you can make it easier to probe particular nodes by changing the view. The following table describes how to manipulate objects in the graphics window:

Action	Using Graphics Toolbar Buttons and the Mouse
Rotate view (vertical, horizontal)	After clicking the <b>Rotate View</b> icon, <b>S</b> , press and hold the left mouse button and drag the mouse. Dragging side to side rotates the view about the vertical axis, and dragging up and down rotates the view about the horizontal axis.
Translate or pan view	After clicking the <b>Pan</b> icon, +++, press and hold the left mouse button and drag the object with the mouse until the view is satisfactory.
Zoom in and out of view	After clicking the <b>Zoom In/Out</b> icon, (I), press and hold the left mouse button and drag the mouse up and down to zoom in and out of the view.
Zoom to selected area	After clicking the <b>Zoom to Area</b> icon, $\textcircled{(\textcircled{)}}$ , press and hold the left mouse button and drag the mouse diagonally to the right. This action will cause a rectangle to appear in the display. When you release the mouse button, a new view will be displayed that consists entirely of the contents of the rectangle. Note that to zoom in, you must drag the mouse to the right, and to zoom out, you must drag the mouse to the left.

**Table 3.1: View Manipulation Instructions** 

Clicking the **Fit to Window** icon, will cause the object to fit exactly and be centered in the window.

After you have clicked a button in the graphics toolbar, you can return to the default

mouse button settings by clicking

To judge the scale of your 3D geometry, you can click the Orthographic Projection icon,

 $\overset{\textcircled{}}{}$ . This will display the length scale ruler near the bottom of the graphics window.

Note that you can change the default mouse button actions in the **Viewing** tab (in the **Mouse** group box). For more information, see the Fluent User's Guide.

2. Manipulate the mesh display using the **Views** dialog box to obtain a front view as shown in Figure 3.2: The Hexahedral Mesh for the Mixing Elbow (p. 130).

In the **Viewing** ribbon tab, locate the **Display** group box.

 Display

 Views...
 Image: Provide the state of the sta

Then click the Views... button to open the Views dialog box.

Viewing →	Display → Views
-----------	-----------------

<b>E</b> Views		<b>—</b>
Views back bottom front isometric left right top Save Name front	Actions Default Auto Scale Previous Save Delete Read Write	Mirror Planes [0/1] symmetry Define Plane Periodic Repeats Define
Арріу	Camera	Close Help

## Note

You can also open the **Views** dialog box from the tree, by expanding **Results**, rightclicking **Graphics**, and selecting **Views...**.

**Results** 
$$\rightarrow$$
 Graphics  $\stackrel{\bullet}{\longrightarrow}$  Views...

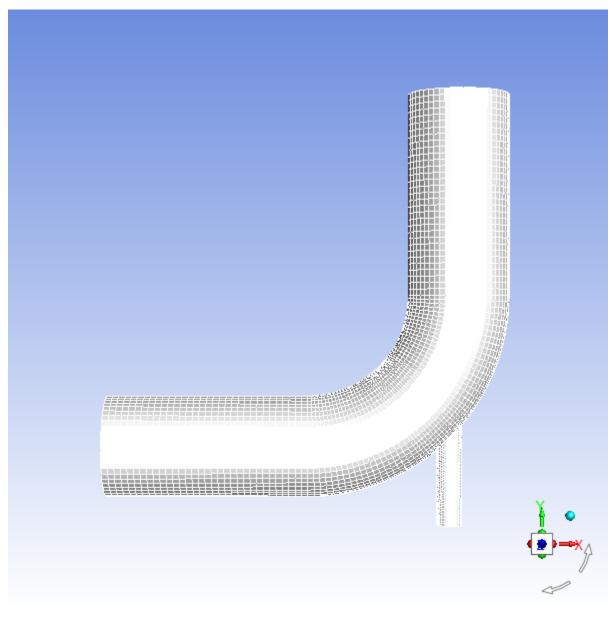
a. Select **front** from the **Views** selection list.

## Note

A list item is selected if it is highlighted, and deselected if it is not highlighted.

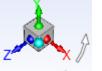
b. Click Apply and close the Views dialog box.





## Extra

You can also change the orientation of the objects in the graphics window using the



~

axis triad

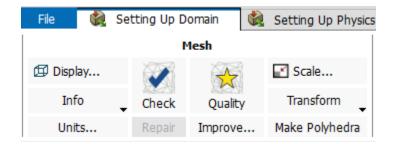
as follows:

- To orient the model in the positive/negative direction, click an axis/semi-sphere.
- To orient the model in the negative/positive direction, right-click an axis/semi-sphere.
- To set the isometric view, click the cyan iso-ball.

- To perform in-plane clockwise or counterclockwise 90° rotations, click the white rotational arrows .
- To perform free rotations in any direction, click and hold—in the vicinity of the triad—and use the mouse. Release the left mouse button to stop rotating.

# 3.4.4. Setting Up Domain

In this step, you will perform the mesh-related activities using the **Setting Up Domain** ribbon tab (**Mesh** group box).



1. Check the mesh.

```
Setting Up Domain \rightarrow Mesh \rightarrow Check
```

ANSYS Fluent will report the results of the mesh check in the console.

```
Domain Extents:
    x-coordinate: min (m) = -8.000000e+00, max (m) = 8.000000e+00
    y-coordinate: min (m) = -9.134633e+00, max (m) = 8.000000e+00
    z-coordinate: min (m) = 0.000000e+00, max (m) = 2.000000e+00
Volume statistics:
    minimum volume (m3): 5.098298e-04
    maximum volume (m3): 2.330736e-02
        total volume (m3): 1.607154e+02
Face area statistics:
    minimum face area (m2): 4.865882e-03
    maximum face area (m2): 1.017924e-01
Checking mesh......
Done.
```

The mesh check will list the minimum and maximum x, y, and z values from the mesh in the default SI unit of meters. It will also report a number of other mesh features that are checked. Any errors in the mesh will be reported at this time. Ensure that the minimum volume is not negative, since ANSYS Fluent cannot begin a calculation when this is the case.

#### Note

The minimum and maximum values may vary slightly when running on different platforms.

2. Scale the mesh.



🞴 Scale M	esh			×
- Domain E	Extents			Scaling
Xmin (in)	-8	Xmax (in)	8	Convert Units
Ymin (in)	-9.134634	Ymax (in)	8	Specify Scaling Factors
Zmin (in)	0	Zmax (in)	2	Mesh Was Created In
View Length Unit In in		Scaling Factors           X         0.0254           Y         0.0254           Z         0.0254           Scale         Unscale		
Close Help				

- a. Ensure that **Convert Units** is selected in the **Scaling** group box.
- b. From the **Mesh Was Created In** drop-down list, select **in** by first clicking the down-arrow button and then clicking the **in** item from the list that appears.
- c. Click **Scale** to scale the mesh.

## Warning

Be sure to click the **Scale** button only once.

**Domain Extents** will continue to be reported in the default SI unit of meters.

- d. Select in from the View Length Unit In drop-down list to set inches as the working unit for length.
- e. Confirm that the domain extents are as shown in the previous dialog box.
- f. Close the Scale Mesh dialog box.

The mesh is now sized correctly and the working unit for length has been set to inches.

#### Note

Because the default SI units will be used for everything except length, there is no need to change any other units in this problem. The choice of inches for the unit of length has been made by the actions you have just taken. If you want a different working unit for length, other than inches (for example, millimeters), click **Units...** in the **Setting Up Domain** ribbon tab (**Mesh** group box) and make the appropriate change in the **Set Units** dialog box.

## 3. Check the mesh.

## Setting Up Domain $\rightarrow$ Mesh $\rightarrow$ Check

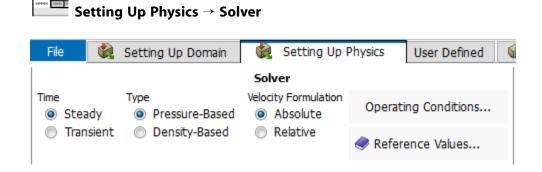
#### Note

It is a good idea to check the mesh after you manipulate it (that is, scale, convert to polyhedra, merge, separate, fuse, add zones, or smooth and swap). This will ensure that the quality of the mesh has not been compromised.

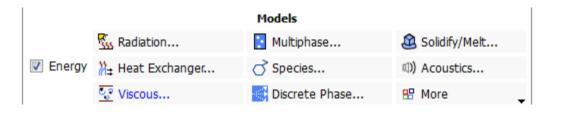
## 3.4.5. Setting Up Physics

In the steps that follow, you will select a solver and specify physical models, material properties, and zone conditions for your simulation using the **Setting Up Physics** ribbon tab.

1. In the **Solver** group box of the **Setting Up Physics** ribbon tab, retain the default selection of the steady pressure-based solver.



2. Set up your models for the CFD simulation using the **Models** group box of the **Setting Up Physics** ribbon tab.



#### Note

You can also use the **Models** task page, which can be accessed from the tree by expanding **Setup** and double-clicking the **Models** tree item.

a. Enable heat transfer by activating the energy equation.

In the Setting Up Physics ribbon tab, enable Energy (Models group box).

# 

#### Note

You can also double-click the **Setup/Models/Energy** tree item and enable the energy equation in the **Energy** dialog box.

b. Enable the k- $\varepsilon$  turbulence model.

 $\blacksquare Setting Up Physics \rightarrow Models \rightarrow Viscous...$ 

Viscous Model		٢.
Model	Model Constants	
Inviscid	Cmu	*
🔘 Laminar	0.09	
<ul> <li>k-epsilon (2 eqn)</li> <li>k-omega (2 eqn)</li> <li>Transition k-kl-omega (3 eqn)</li> </ul>	C1-Epsilon	
	1.44	
	C2-Epsilon	
	1.92	=
<ul> <li>Reynolds Stress (7 eqn)</li> </ul>	TKE Prandtl Number	
Scale-Adaptive Simulation (SAS)	1	
Detached Eddy Simulation (DES)	TDR Prandtl Number	
Carge Eddy Simulation (LES)	1.3	
k-epsilon Model	Energy Prandtl Number	
<ul> <li>Standard</li> </ul>	0.85	
C RNG	Wall Prandtl Number	_
Realizable		Ť
Near-Wall Treatment          Standard Wall Functions         Scalable Wall Functions         Non-Equilibrium Wall Functions         Enhanced Wall Treatment         Menter-Lechner         User-Defined Wall Functions         Enhanced Wall Treatment Options         Pressure Gradient Effects         Thermal Effects         Options         Viscous Heating         Curvature Correction         Production Kato-Launder         Production Limiter	User-Defined Functions Turbulent Viscosity none Prandtl Numbers TKE Prandtl Number none TDR Prandtl Number none Energy Prandtl Number None Wall Prandtl Number None	
OK (	Cancel Help	at

i. Select **k-epsilon** from the **Model** list.

The **Viscous Model** dialog box will expand.

ii. Retain the default selection of **Standard** in the **k-epsilon Model** group box.

iii. Select Enhanced Wall Treatment in the Near-Wall Treatment group box.

#### Note

The default Standard Wall Functions are generally applicable if the first cell center adjacent to the wall has a y+ larger than 30. In contrast, the Enhanced Wall Treatment option provides consistent solutions for all y+ values. Enhanced Wall Treatment is recommended when using the k-epsilon model for general single-phase fluid flow problems. For more information about Near Wall Treatments in the k-epsilon model, refer to the Fluent User's Guide.

iv. Click **OK** to accept all the other default settings and close the **Viscous Model** dialog box.

Note that the **Viscous...** label in the ribbon is displayed in blue to indicate that the Viscous model is enabled. Also **Energy** and **Viscous** appear as enabled under the **Setup/Models** tree branch.

#### Note

While the ribbon is the primary tool for setting up and solving your problem, the tree is a dynamic representation of your case. The models, materials, conditions, and other settings that you have specified in your problem will appear in the tree. Many of the frequently used ribbon items are also available via the right-click functionality of the tree.

3. Set up the materials for the CFD simulation using the **Materials** group box of the **Setting Up Physics** ribbon tab.



Create a new material called water using the Create/Edit Materials dialog box.

a. In the Setting Up Physics ribbon tab, click Create/Edit... (Materials group box).

Setting Up Physics  $\rightarrow$  Materials  $\rightarrow$  Create/Edit...

Create/Edit Materials		<b>—</b>
Name	Material Type	Order Materials by
water	fluid	<ul> <li>Name</li> </ul>
Chemical Formula	Fluent Fluid Materials	Chemical Formula
	water	Fluent Database
	Mixture	User-Defined Database
	none	VSer-Denned Database
Properties		
Cp (Specific Heat) (j/kg-k) constant	▼ [Edit] ^	
4216		
Thermal Conductivity (w/m-k) constant	▼ [Edit	
0.677	=	
Viscosity (kg/m-s) constant	- Edit	
0.0008		
	*	
	Change/Create Delete Close Help	

- b. Enter water for Name.
- c. Enter the following values in the **Properties** group box:

Property	Value
Density	1000 [kg/m <sup>3</sup> ]
$c_p$ (Specific Heat)	4216 [J/kg-K]
Thermal Conductivity	0.677 [W/m-K]
Viscosity	8e-04 [kg/m-s]

#### d. Click Change/Create.

A **Question** dialog box will open, asking if you want to overwrite air. Click **No** so that the new material **water** is added to the **Fluent Fluid Materials** list of materials that originally contained only **air**.

Question		×
?	Change/Create mixture and Overwrite air?	
	Yes	

## Extra

You could have copied the material **water-liquid** (h2o < l >) from the materials database (accessed by clicking the **Fluent Database...** button). If the properties in the database are different from those you want to use, you can edit the values in the

**Properties** group box in the **Create/Edit Materials** dialog box and click **Change/Create** to update your local copy. The original copy will not be affected.

e. Ensure that there are now two materials (water and air) defined locally by examining the **Fluent Fluid Materials** drop-down list.

Both the materials will also be listed under **Fluid** in the **Materials** task page and under the **Materials** tree branch.

- f. Close the Create/Edit Materials dialog box.
- 4. Set up the cell zone conditions for the fluid zone (**fluid**) using the **Zones** group box of the **Setting Up Physics** ribbon tab.



a. In the Setting Up Physics tab, click Cell Zones (Zones group box).



This opens the **Cell Zone Conditions** task page.

Task Page ×
Cell Zone Conditions
Zone Filter Text
fluid
Phase Type ID
· · · · · · · · · · · · · · · · · · ·
Edit Copy Profiles
Parameters Operating Conditions
Display Mesh
Porous Formulation
Superficial Velocity     Advantage
O Physical Velocity
Help

b. Double-click **fluid** in the **Zone** list to open the **Fluid** dialog box.

Introduction to Using ANSYS Fluent: Fluid Flow and Heat Transfer in a Mixing Elbow

E Fluid								
Zone Name								<b>—</b> X—
fluid								
Material Name water		▼ Edit						
Frame Motion								
Mesh Motion	Laminar Zone 📗	Fixed Values						
Porous Zone								
Reference Frame	Mesh Motion	Porous Zone	3D Fan Zone	Embedded LES	Reaction	Source Terms	Fixed Values	Multiphase
Rotation-Axis Origi	'n		Rotation-Axis Di	rection				
X (in) 0	constant	•	X 0	constant	•			
Y (in) 0	constant	•)	Y 0	constant 👻				
Z (in) 0	constant	•	Z 1	constant	•			
			ОК	Cancel Help				.H

## Note

You can also double-click the **Setup/Cell Zone Conditions/fluid** tree item in order to open the corresponding dialog box.

- c. Select water from the Material Name drop-down list.
- d. Click OK to close the Fluid dialog box.
- 5. Set up the boundary conditions for the inlets, outlet, and walls for your CFD analysis using the **Zones** group box of the **Setting Up Physics** ribbon tab.

Zones
Cell Zones
Boundaries
Profiles

a. In the Setting Up Physics tab, click Boundaries (Zones group box).

Setting Up Physics  $\rightarrow$  Zones  $\rightarrow$  Boundaries

This opens the **Boundary Conditions** task page where the boundaries defined in your simulation are displayed in the **Zone** selection list.

Task Page ×
Boundary Conditions
Zone Filter Text
<ul> <li>Inlet         velocity-inlet-5         velocity-inlet-6</li> <li>Internal         default-interior</li> <li>Outlet         pressure-outlet-7</li> <li>Symmetry         symmetry         symmetry</li> <li>Wall         wall</li> </ul>
Phase Type ID mixture -1
Edit       Copy       Profiles         Parameters       Operating Conditions         Display Mesh       Periodic Conditions
Highlight Zone
Help

## Note

To display boundary zones grouped by zone type (as shown previously), click the

**Toggle Tree View** button (F) in the upper right corner of the **Boundary Conditions** task page and select **Zone Type** under **Group By**.

Here the zones have names with numerical identifying tags. It is good practice to give boundaries meaningful names in a meshing application to help when you set up the model. You can also change boundary names in Fluent by simply editing the boundary and making revisions in the **Zone Name** text box.

b. Set the boundary conditions at the cold inlet (**velocity-inlet-5**).

## Tip

If you are unsure of which inlet zone corresponds to the cold inlet, you can probe the mesh display using the right mouse button or the probe toolbar button

( ) as described previously in this tutorial. The information will be displayed in the ANSYS Fluent console, and the zone you probed will be automatically selected from the **Zone** selection list in the **Boundary Conditions** task page.

i. Double-click velocity-inlet-5 to open the Velocity Inlet dialog box.

1	Velocity Inlet	t							×
	Zone Name								
	velocity-inlet-5								
	Momentum	Thermal	Radiation	Species	DPM	Multip	hase	Potential	UDS
	Velocity	y Specificatior	Method Mag	gnitude, No	rmal to B	oundary			•
		Referen	ce Frame Abs	solute					•
		Velocity Ma	gnitude (m/s)	0.4			const	ant	-
	Supersonic/Init	tial Gauge Pre	ssure (pascal)	0			const	ant	•
		- Turbulence							
		Specification	Method Inte	nsity and H	ydraulic D	)iameter			<b>-</b>
				Turbulent I	Intensity	(%) 5			Р
				Hydraulic	Diameter	(in) 4			P
	OK Cancel Help								

- ii. Retain the default selection of **Magnitude**, **Normal to Boundary** from the **Velocity Specification Method** drop-down list.
- iii. Enter 0.4 [m/s] for Velocity Magnitude.
- iv. In the **Turbulence** group box, select **Intensity and Hydraulic Diameter** from the **Specification Method** drop-down list.
- v. Retain the default value of 5 [%] for **Turbulent Intensity**.
- vi. Enter 4 [inches] for Hydraulic Diameter.

The hydraulic diameter  $D_h$  is defined as:

$$D_h = \frac{4A}{P_w}$$

where A is the cross-sectional area and  $P_w$  is the wetted perimeter.

vii. Click the **Thermal** tab.

Velocity Inlet						×
Zone Name						
velocity-inlet-5						
Momentum Thermal	Radiation	Species	DPM	Multiphase	Potential	UDS
Temperature (k) 293.15		consta	nt	•		
	OI	Cancel	Help			

viii.Enter 293.15 [K] for Temperature.

ix. Click **OK** to close the **Velocity Inlet** dialog box.

## Note

You can also access the **Velocity Inlet** dialog box by double-clicking the **Setup/Boundary Conditions/velocity-inlet-5** tree item.

c. In a similar manner, set the boundary conditions at the hot inlet (**velocity-inlet-6**), using the values in the following table:

Setting	Value
Velocity Specification Method	Magnitude, Normal to Boundary
Velocity Magnitude	1.2 [m/s]
Specification Method	Intensity and Hydraulic Diameter
Turbulent Intensity	5 [%]
Hydraulic Diameter	1 [inch]
Temperature	313.15 [K]

d. Double-click **pressure-outlet-7** in the **Zone** selection list and set the boundary conditions at the outlet, as shown in the following figure.

Pressure Outle	t						<b>—</b>
Zone Name							
pressure-outlet-7	7						
Momentum	Thermal	Radiation	Species	DPM	Multiphase	Potential	UDS
Ba	ckflow Refe	rence Frame	Absolute				•
	Gauge P	ressure (pas	cal) 0		cor	nstant	
Backflow Direction	on Specifica	tion Method	Normal to B	oundary			•
Backflo	w Pressure	Specification	Total Pressu	ire			•
🔲 Radial Equilibr	Radial Equilibrium Pressure Distribution						
🔲 Average Pres	ssure Specifi	ication					
Target Mass I	Flow Rate						
	- Turbulen	се					
	Specificat	ion Method 🛛	Intensity and	d Hydraul	ic Diameter		•
		Backfl	ow Turbulen	t Intensi	ty (%) 5		P
	Backflow Hydraulic Diameter (in) 4						
			OK Cance	Help	]		

## Note

- You do not need to set a backflow temperature in this case (in the **Thermal** tab) because the material properties are not functions of temperature. If they were, a flow-weighted average of the inlet conditions would be a good starting value.
- ANSYS Fluent will use the backflow conditions only if the fluid is flowing into the computational domain through the outlet. Since backflow might occur at some point during the solution procedure, you should set reasonable backflow conditions to prevent convergence from being adversely affected.
- e. For the wall of the pipe (wall), retain the default value of 0 W/m<sup>2</sup> for Heat Flux in the Thermal tab.

🛃 Wall		<b>×</b>
Zone Name wall		
Adjacent Cell Zone fluid		
Momentum Thermal	Radiation Species DPM Multiphase UDS Wall Film Potential	
Thermal Conditions		
<ul> <li>Heat Flux</li> <li>Temperature</li> <li>Convection</li> <li>Radiation</li> <li>Mixed</li> <li>via System Coupling</li> <li>via Mapped Interface</li> </ul>		▼ ₽ ▼ Edit
Material Name aluminum	Edit	
	OK Cancel Help	

# 3.4.6. Solving

In the steps that follow, you will set up and run the calculation using the **Solving** ribbon tab.

## Note

You can also use the task pages listed under the **Solution** tree branch to perform solution-related activities.

- 1. Select a solver scheme.
  - a. In the **Solving** ribbon tab, click **Methods...** (Solution group box).





Task Page ×
Solution Methods
Pressure-Velocity Coupling
Scheme
SIMPLE
Spatial Discretization
Gradient
Least Squares Cell Based
Pressure
Second Order
Momentum
Second Order Upwind
Turbulent Kinetic Energy
First Order Upwind
Turbulent Dissipation Rate
First Order Upwind
Transient Formulation
· · · · · · · · · · · · · · · · · · ·
Non-Iterative Time Advancement
Frozen Flux Formulation
Pseudo Transient
Warped-Face Gradient Correction
High Order Term Relaxation Options
Default
Help

- b. In the Solution Methods task page, retain the default selections for the Scheme and Spatial Discretization.
- 2. Enable the plotting of residuals during the calculation.
  - a. In the **Solving** ribbon tab, click **Residuals...** (**Reports** group box).

Residuals Convergence
Definitions File Plot

Residual Monitors					×		
Options	Equations						
Print to Console	Residual	Monitor	Check Converge	ence Absolute Criteria	<u> </u>		
V Plot	continuity	<b>V</b>	$\checkmark$	0.001	E		
Window	x-velocity	<b>V</b>		0.001			
1 🚖 Curves Axes	y-velocity	<b>V</b>		0.001			
Iterations to Plot	z-velocity	<b>V</b>		0.001			
1000				- ac			
	Residual Values			Convergence Criterion			
	Normalize		Iterations	absolute	•		
Iterations to Store			5				
1000 🚔	Scale			Convergence Conditi	ons		
	Compute Loca	l Scale					
ОК	Plot Renormaliz	car	Help				
					н		

## Note

You can also access the **Residual Monitors** dialog box by double-clicking the **Solution/Monitors/Residual** tree item.

- b. Ensure that **Plot** is enabled in the **Options** group box.
- c. Retain the default value of 0.001 for the Absolute Criteria of continuity.
- d. Click OK to close the Residual Monitors dialog box.

### Note

By default, the residuals of all of the equations solved for the physical models enabled for your case will be monitored and checked by ANSYS Fluent as a means to determine the convergence of the solution. It is a good practice to also create and plot a surface report definition that can help evaluate whether the solution is truly converged. You will do this in the next step.

3. Create a surface report definition of average temperature at the outlet (pressure-outlet-7).

Solving  $\rightarrow$  Reports  $\rightarrow$  Definitions  $\rightarrow$  New  $\rightarrow$  Surface Report  $\rightarrow$  Mass-Weighted Average...

Surface Report Definition	
Name	Report Type
outlet-temp-avg	Mass-Weighted Average
Options	Custom Vectors
	Vectors of
Per Surface	▼
Average Over	Custom Vectors
1	Field Variable
Report Files [0/0]	Temperature
	Static Temperature 👻
	Surfaces Filter Text
	default-interior
	pressure-outlet-7
Report Plots [0/0]	symmetry velocity-inlet-5
	velocity-inlet-6
	wall
Create	
Report File	
Report Plot	
Frequency 3	
Print to Console	Highlight Surfaces
Create Output Parameter	New Surface 🔻
ОК	Compute Cancel Help

### Note

You can also access the **Surface Report Definition** dialog box by right-clicking **Report Definitions** in the tree (under **Solution**) and selecting **New/Surface Report/Mass-Weighted Average...** from the menu that opens.

- a. Enter outlet-temp-avg for the Name of the report definition.
- b. Enable Report File, Report Plot, and Print to Console in the Create group box.

During a solution run, ANSYS Fluent will write solution convergence data in a report file, plot the solution convergence history in a graphics window, and print the value of the report definition to the console.

c. Set **Frequency** to 3 by clicking the up-arrow button.

This setting instructs ANSYS Fluent to update the plot of the surface report, write data to a file, and print data in the console after every 3 iterations during the solution.

- d. Select Temperature... and Static Temperature from the Field Variable drop-down lists.
- e. Select **pressure-outlet-7** from the **Surfaces** selection list.
- f. Click OK to save the surface report definition and close the Surface Report Definition dialog box.

The new surface report definition **outlet-temp-avg** will appear under the **Solution/Report Definitions** tree item. ANSYS Fluent also automatically creates the following items:

- outlet-temp-avg-rfile (under the Solution/Monitors/Report Files tree branch)
- outlet-temp-avg-rplot (under the Solution/Monitors/Report Plots tree branch)
- 4. In the tree, double-click **outlet-temp-avg-rfile** (under **Solution/Monitors/Report Files**) and examine the report file settings in the **Edit Report File** dialog box.

Edit Report File			<b>—</b>
Name			
outlet-temp-avg-rfile			
Available Report Definitions [0/0]	- <u>x</u>	Selected Report Definitions [0/1]	x
		outlet-temp-avg	
	Add>>		
	< <remove< td=""><td></td><td></td></remove<>		
Output File Base Name			
\\outlet-temp-avg-rfile.out	0	New - Edit	
Full File Name			
Get Data Every			
3 🗢 Iteration	•		
Print to Con:	sole		
	OK Cancel	Help	

The dialog box is automatically populated with data from the **outlet-temp-avg** report definition.

a. Verify that outlet-temp-avg is in the Selected Report Definitions list.

If you had created multiple report definitions, the additional ones would be listed under **Available Report Definitions**, and you could use the **Add**>> and **<<Remove** buttons to manage which were written in this particular report definition file.

b. (optional) Edit the name and location of the resulting file as necessary using the **Output File Base Name** field or **Browse...** button.

- c. Click **OK** to close the **Edit Report File** dialog box.
- 5. Create a convergence condition for **outlet-temp-avg**.

Solving → Reports → Convergence	•••
---------------------------------	-----

Convergence Conditions						×
Conditions De	port finition utlet-temp-a 🔻	Stop Criterion 1e-5	Ignore Iterations Bef 20	Use fore Iterations 15	Print	Delete
Add Choose Condition All Conditions are Met Any Condition is Met	Every Iteration		Residuals			
OK Cancel Help						

- a. Click the **Add** button.
- b. Enter con-outlet-temp-avg for Conditions.
- c. Select outlet-temp-avg from the Report Definition drop-down list.
- d. Enter 1e-5 for **Stop Criterion**.
- e. Enter 20 for Ignore Iterations Before.
- f. Enter 15 for Use Iterations.
- g. Enable **Print**.
- h. Set Every Iteration to 3.
- i. Click **OK** to save the convergence condition settings and close the **Convergence Conditions** dialog box.

These settings will cause Fluent to consider the solution converged when the surface report definition value for each of the previous 15 iterations is within 0.001% of the current value. Convergence of the values will be checked every 3 iterations. The first 20 iterations will be ignored, allowing for any initial solution dynamics to settle out. Note that the value printed to the console is the deviation between the current and previous iteration values only.

6. Initialize the flow field using the **Initialization** group box of the **Solving** ribbon tab.

Solving $\rightarrow$ Initialization					
	Initialization				
	Method		Patch		
	O Hybrid	More Settings	Reset Statistics		
	Standard	Options		t = 0	
			Reset DPM	Initialize	

- a. Retain the default selection of Hybrid from the Method list.
- b. Click **Initialize**.
- 7. Save the case file (elbow1.cas.gz).



Select File	? 🔀
Look in: 🕞 C:\Tutorials\introduction 🔹 🔾 🔾 🖓	📑 🗉 🔳
My Com Docume Name Solution_files	
Case File elbow1.cas.gz	ОК
Files of type: Case Files (*.cas*)	Cancel
Filter String	Filter
Vrite Binary Files	

a. (optional) Indicate the folder in which you would like the file to be saved.

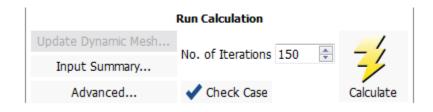
By default, the file will be saved in the folder from which you read in elbow.msh (that is, the introduction folder). You can indicate a different folder by browsing to it or by creating a new folder.

b. Enter elbow1.cas.gz for Case File.

Adding the extension .gz to the end of the file name extension instructs ANSYS Fluent to save the file in a compressed format. You do not have to include .cas in the extension (for example, if you enter elbow1.gz, ANSYS Fluent will automatically save the file as elbow1.cas.gz). The .gz extension can also be used to save data files in a compressed format.

- c. Ensure that the default Write Binary Files option is enabled, so that a binary file will be written.
- d. Click **OK** to save the case file and close the **Select File** dialog box.
- 8. Start the calculation by requesting 150 iterations in the **Solving** ribbon tab (**Run Calculation** group box).

## Solving → Run Calculation



- a. Enter 150 for No. of Iterations.
- b. Click Calculate.

### Note

By starting the calculation, you are also starting to save the surface report data at the rate specified in the **Surface Report Definition** dialog box. If a file already exists in your working directory with the name you specified in the **Edit Report File** dialog box, then a **Question** dialog box will open, asking if you would like to append the new data to the existing file. Click **No** in the **Question** dialog box, and then click **OK** in the **Warning** dialog box that follows to overwrite the existing file.

As the calculation progresses, the surface report history will be plotted in the **outlet-temp-avgrplot** tab in the graphics window (Figure 3.3: Convergence History of the Mass-Weighted Average Temperature (p. 153)).

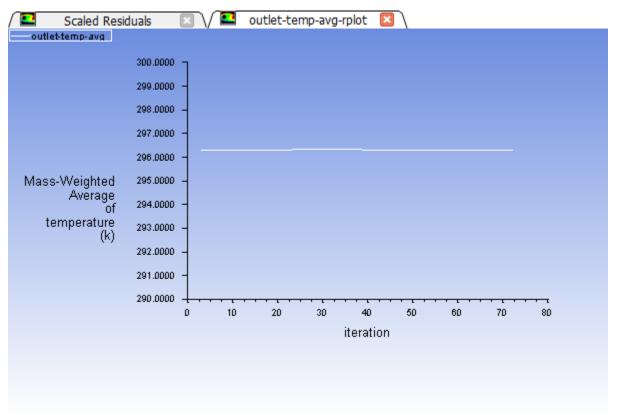
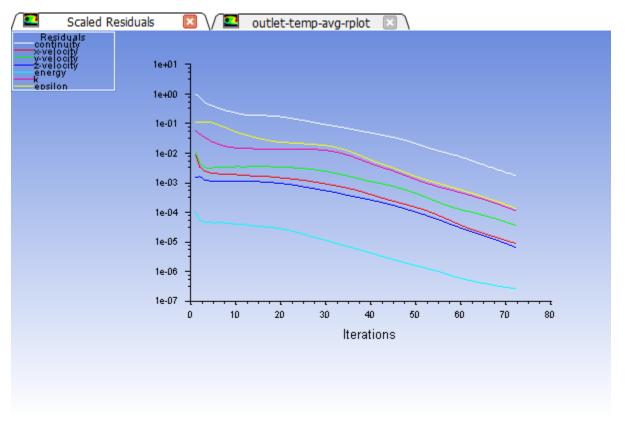


Figure 3.3: Convergence History of the Mass-Weighted Average Temperature

Similarly, the residuals history will be plotted in the **Scaled Residuals** tab in the graphics window (Figure 3.4: Residuals (p. 154)).



#### Figure 3.4: Residuals

### Note

You can monitor the two convergence plots simultaneously by right-clicking a tab in the graphics window and selecting **SubWindow View** from the menu that opens. To return to a tabbed graphics window view, right-click a graphics window title area and select **Tabbed View**.

Since the residual values vary slightly by platform, the plot that appears on your screen may not be exactly the same as the one shown here.

The solution will be stopped by ANSYS Fluent when any of the following occur:

- the surface report definition converges to within the tolerance specified in the **Convergence Conditions** dialog box
- the residual monitors converge to within the tolerances specified in the **Residual Monitors** dialog box
- the number of iterations you requested in the **Run Calculation** task page has been reached

In this case, the solution is stopped when the convergence criterion on outlet temperature is satisfied, after approximately 72 iterations. The exact number of iterations for convergence will vary, depending on the platform being used. An **Information** dialog box will open to alert you that the calculation is complete. Click **OK** in the **Information** dialog box to proceed.

9. Examine the plots for convergence (Figure 3.3: Convergence History of the Mass-Weighted Average Temperature (p. 153) and Figure 3.4: Residuals (p. 154)).

## Note

There are no universal metrics for judging convergence. Residual definitions that are useful for one class of problem are sometimes misleading for other classes of problems. Therefore it is a good idea to judge convergence not only by examining residual levels, but also by monitoring relevant integrated quantities and checking for mass and energy balances.

There are three indicators that convergence has been reached:

• The residuals have decreased to a sufficient degree.

The solution has converged when the **Convergence Criterion** for each variable has been reached. The default criterion is that each residual will be reduced to a value of less than  $10^{-3}$ , except the **energy** residual, for which the default criterion is  $10^{-6}$ .

• The solution no longer changes with more iterations.

Sometimes the residuals may not fall below the convergence criterion set in the case setup. However, monitoring the representative flow variables through iterations may show that the residuals have stagnated and do not change with further iterations. This could also be considered as convergence.

• The overall mass, momentum, energy, and scalar balances are obtained.

You can examine the overall mass, momentum, energy and scalar balances in the **Flux Reports** dialog box. The net imbalance should be less than 0.2 % of the net flux through the domain when the solution has converged. In the next step you will check to see if the mass balance indicates convergence.

10. Examine the mass flux report for convergence using the **Postprocessing** ribbon tab.



Introduction to Using ANSYS Fluent: Fluid Flow and Heat Transfer in a Mixing Elbow

Flux Reports		
Options Mass Flow Rate	Boundaries Filter Text	Results
<ul> <li>Total Heat Transfer Rate</li> <li>Radiation Heat Transfer Rate</li> </ul>	default-interior pressure-outlet-7	-1.919306039810181
	symmetry velocity-inlet-5 velocity-inlet-6	1.617521166801453 0.3018067181110382
	wall	
	4 4	
Save Output Parameter		Net Results (kg/s) 2.18451e-05
	Compute Write Close Help	2.104516-05
	Compute Write Close Help	

- a. Ensure that Mass Flow Rate is selected from the Options list.
- b. Select pressure-outlet-7, velocity-inlet-5, and velocity-inlet-6 from the Boundaries selection list.
- c. Click Compute.

The individual and net results of the computation will be displayed in the **Results** and **Net Results** boxes, respectively, in the **Flux Reports** dialog box, as well as in the console.

The sum of the flux for the inlets should be very close to the sum of the flux for the outlets. The net results show that the imbalance in this case is well below the 0.2% criterion suggested previously.

- d. Close the Flux Reports dialog box.
- 11. Save the data file (elbow1.dat.gz).

```
File \rightarrow Write \rightarrow Data...
```

In later steps of this tutorial you will save additional case and data files with different suffixes.

# 3.4.7. Displaying the Preliminary Solution

In the steps that follow, you will visualize various aspects of the flow for the preliminary solution using the **Postprocessing** ribbon tab.

1. Display filled contours of velocity magnitude on the symmetry plane (Figure 3.5: Predicted Velocity Distribution after the Initial Calculation (p. 158)).

Postprocessing  $\rightarrow$  Graphics  $\rightarrow$  Contours  $\rightarrow$  Edit...

Contours		×	
Options	Contours of		
<ul> <li>Filled</li> <li>Node Values</li> </ul>	Velocity		
Global Range	Velocity Magnitude	•	
Auto Range	Min	Max	
Clip to Range	0	0	
Draw Profiles	Surfaces Filter Text		
Coloring Banded Smooth Levels Setup 20 1	default-interior pressure-outlet-7		
	symmetry velocity-inlet-5 velocity-inlet-6		
	wall z=0_outlet		
	New Surface 🔻		
	Display Compute	e Close Help	

- a. Enable **Filled** in the **Options** group box.
- b. Ensure that Node Values is enabled in the Options group box.
- c. Select Velocity... and Velocity Magnitude from the Contours of drop-down lists.
- d. Select symmetry from the Surfaces selection list.
- e. Click Display to display the contours in the active graphics window. Clicking the Fit to Window icon

( ) will cause the object to fit exactly and be centered in the window.

#### Note

If you cannot see the velocity contour display, select the appropriate tab in the graphics window.

- f. Close the **Contours** dialog box.
- g. Disable the Lighting option in the Viewing ribbon tab (in the Display group box).

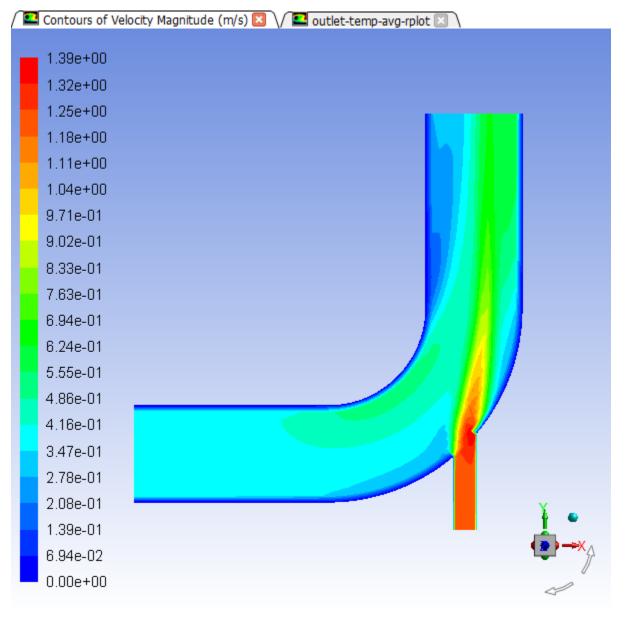


Figure 3.5: Predicted Velocity Distribution after the Initial Calculation

## Extra

When you probe a point in the displayed domain with the right mouse button or the probe tool, the level of the corresponding contour is highlighted in the colormap in the graphics window, and is also reported in the console.

2. Create and display a definition for temperature contours on the symmetry plane (Figure 3.6: Predicted Temperature Distribution after the Initial Calculation (p. 160)).

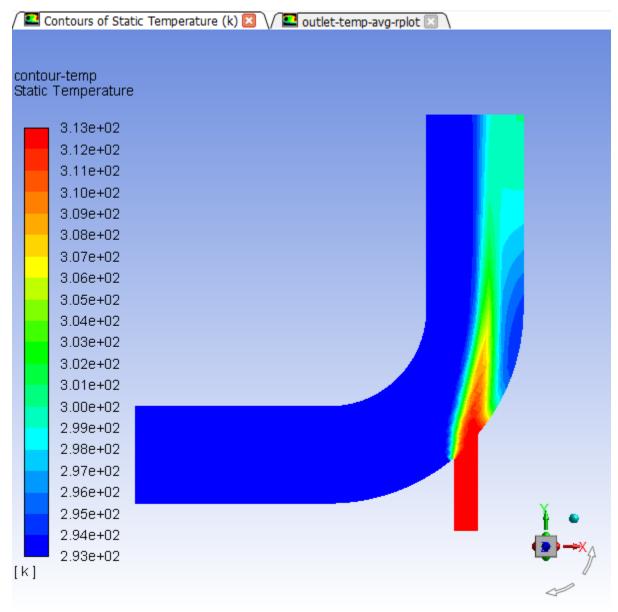
## Postprocessing $\rightarrow$ Graphics $\rightarrow$ Contours $\rightarrow$ New...

You can create contour definitions and save them for later use.

Contours			
Contour Name			
contour-temp			
Options	Contours of		
Filled	Temperature 🔻		
Node Values	Static Temperature 🗸		
<ul> <li>Global Range</li> <li>Auto Range</li> </ul>	Min Max		
Clip to Range	0 0		
Draw Profiles	Surfaces Filter Text		
	default-interior pressure-outlet-7		
Coloring	symmetry		
Banded     Group ath	velocity-inlet-5		
Smooth	velocity-inlet-6		
Colormap Options	wall		
	New Surface 🔻		
Save/Display Compute Close Help			

- a. Enter contour-temp for Contour Name.
- b. Select Temperature... and Static Temperature from the Contours of drop-down lists.
- c. Click Save/Display and close the Contours dialog box.

The new **contour-temp** definition appears under the **Results/Graphics/Contours** tree branch. To edit your contour definition, right-click it and select **Edit...** from the menu that opens.



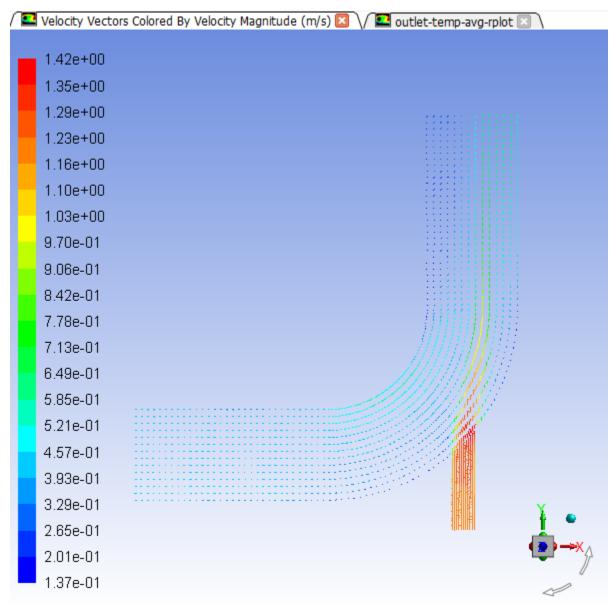
## Figure 3.6: Predicted Temperature Distribution after the Initial Calculation

3. Display velocity vectors on the symmetry plane (Figure 3.9: Magnified View of Resized Velocity Vectors (p. 164)).

Postprocessing  $\rightarrow$  Graphics  $\rightarrow$  Vectors  $\rightarrow$  Edit...

Vectors		
Options Global Range Auto Range Clip to Range	Vectors of Velocity Color by Velocity	▼
<ul> <li>Auto Scale</li> <li>Draw Mesh</li> </ul>	Velocity Magnitude	▼
Style arrow	Min (m/s) 0.1365518	Max (m/s) 1.418624
Scale Skip	Surfaces Filter Text	
4 2 🜩	default-interior pressure-outlet-7	
Custom Vectors	symmetry velocity-inlet-5 velocity-inlet-6 wall	
	New Surface 🔻	
	Display Compute	Close Help

- a. Select **symmetry** from the **Surfaces** selection list.
- b. Click **Display** to plot the velocity vectors.





The **Auto Scale** option is enabled by default in the **Options** group box. This scaling sometimes creates vectors that are too small or too large in the majority of the domain. You can improve the clarity by adjusting the **Scale** and **Skip** settings, thereby changing the size and number of the vectors when they are displayed.

- c. Enter 4 for **Scale**.
- d. Set Skip to 2.
- e. Click **Display** again to redisplay the vectors.

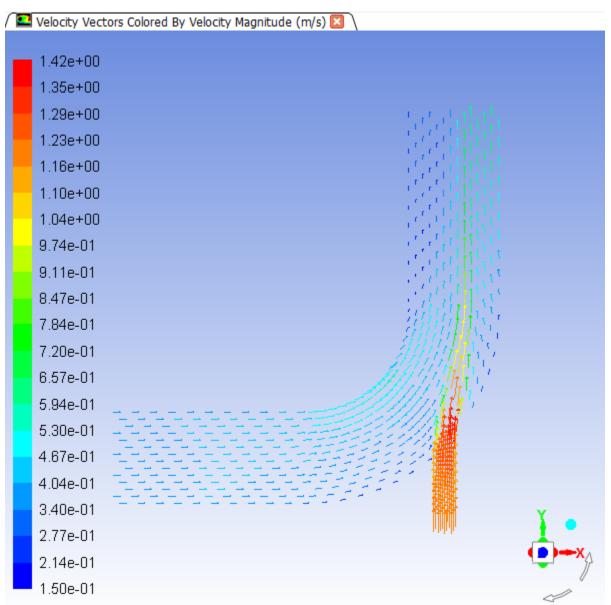
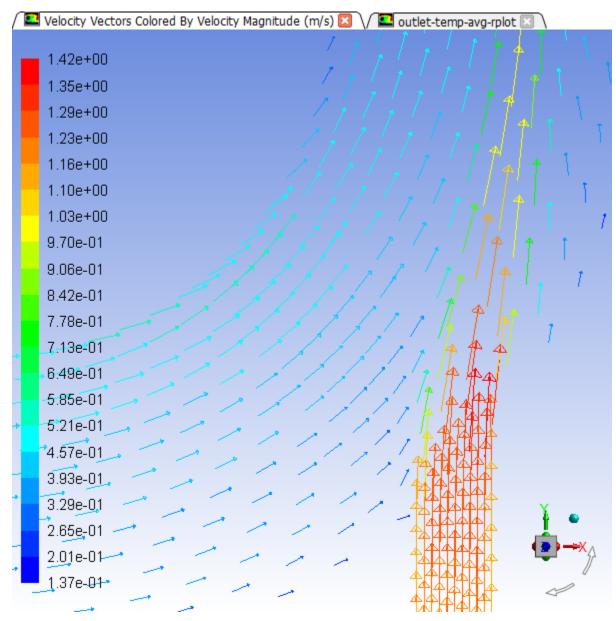


Figure 3.8: Resized Velocity Vectors

- f. Close the **Vectors** dialog box.
- g. Zoom in on the vectors in the display.

To manipulate the image, refer to Table 3.1: View Manipulation Instructions (p. 128). The image will be redisplayed at a higher magnification (Figure 3.9: Magnified View of Resized Velocity Vectors (p. 164)).



#### Figure 3.9: Magnified View of Resized Velocity Vectors

h. Zoom out to the original view.

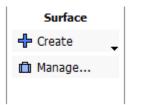
You also have the option of selecting the original view in the **Views** dialog box:

## $\blacksquare \forall Viewing \rightarrow Display \rightarrow Views...$

Select front from the Views selection list and click Apply, then close the Views dialog box.

/iews	Actions Mirror Planes [0/1]			
back	Default			
bottom	Auto Scale symmetry			
front	Previous			
isometric	Save			
left				
right	Delete			
top	Read Define Plane			
	Write Periodic Repeats			
Save Name	Define			
front	Dennem			
Apply Camera Close Help				

4. Create a line at the centerline of the outlet. For this task, you will use the **Surface** group box of the **Postprocessing** tab.



Postprocessing  $\rightarrow$  Surface  $\rightarrow$  Create  $\rightarrow$  Iso-Surface...

Iso-Surface		
Surface of Constant Mesh	•	From Surface Filter Text
Z-Coordinate	•	
Min (in)	Max (in)	pressure-outlet-7
0	2	symmetry velocity-inlet-5
Iso-Values (in)		velocity-inlet-6
0		wall
◄ [		From Zones Filter Text
New Surface Name		fluid
z=0_outlet		
	Create Compute	
		h.

- a. Select Mesh... and Z-Coordinate from the Surface of Constant drop-down lists.
- b. Click **Compute** to obtain the extent of the mesh in the z-direction.

The range of values in the z-direction is displayed in the **Min** and **Max** fields.

- c. Retain the default value of 0 inches for Iso-Values.
- d. Select pressure-outlet-7 from the From Surface selection list.
- e. Enter z=0\_outlet for New Surface Name.
- f. Click Create.

The new line surface representing the intersection of the plane z=0 and the surface pressureoutlet-7 is created, and its name **z=0\_outlet** appears in the **From Surface** selection list.

#### Note

- After the line surface **z=0\_outlet** is created, a new entry will automatically be generated for **New Surface Name**, in case you would like to create another surface.
- If you want to delete or otherwise manipulate any surfaces, click **Manage...** to open the **Surfaces** dialog box.
- g. Close the **Iso-Surface** dialog box.
- 5. Display and save an XY plot of the temperature profile across the centerline of the outlet for the initial solution (Figure 3.10: Outlet Temperature Profile for the Initial Solution (p. 168)).

Postprocessing  $\rightarrow$  Plots  $\rightarrow$  XY Plot  $\rightarrow$  Edit...

Solution XY Plot			
Options Vode Values Vode Vode Values Vode Vode Values Vode Values Vode Vode Vode Values Vode Vode Vode Vode Vode Vode Vode Vode		Plot Direction X 1 Y 0 Z 0	Y Axis Function Temperature Static Temperature X Axis Function Direction Vector Surfaces Filter Text Gefault-interior pressure-outlet-7 symmetry velocity-inlet-5 velocity-inlet-5 velocity-inlet-6 wall z=0_outlet New Surface
Plot Axes Curves Close Help			

- a. Select Temperature... and Static Temperature from the Y Axis Function drop-down lists.
- b. Select the **z=0\_outlet** surface you just created from the **Surfaces** selection list.
- c. Click Plot.
- d. Enable Write to File in the Options group box.

The button that was originally labeled **Plot** will change to **Write...**.

- e. Click Write....
  - i. In the Select File dialog box, enter outlet\_temp1.xy for XY File.
  - ii. Click OK to save the temperature data and close the Select File dialog box.
- f. Close the **Solution XY Plot** dialog box.

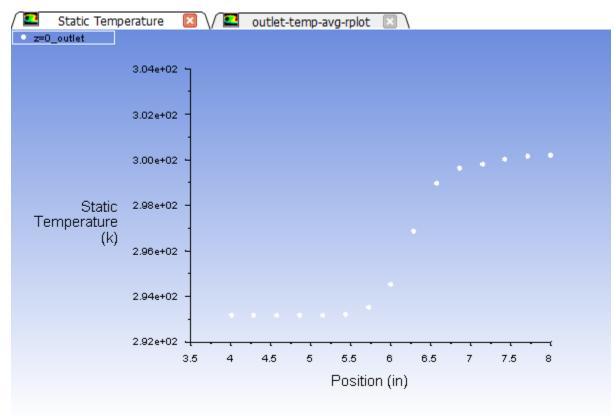


Figure 3.10: Outlet Temperature Profile for the Initial Solution

- 6. Define a custom field function for the dynamic head formula  $(\rho \cdot |V|^2/2)$ .
  - User Defined  $\rightarrow$  Field Functions  $\rightarrow$  Custom...

2	Custom Field Function Calculator							
	finition	0.2.1.2						
	ensity *  V	~ 2/ 2					Select Operand Field Functions from	
	+	-	X		y^x	ABS	Field Functions	
	INV	sin	COS	tan	h	log10	Velocity	-
	0	1	2	3	4	SQRT	Velocity Magnitude	
	5	6	7	8	9	CE/C		
	(	)	PI	e		DEL	Select	
Ne	New Function Name dynamic-head Define Manage Close Help							
				Denn	e manage.			

- a. Select **Density...** and **Density** from the **Field Functions** drop-down lists, and click the **Select** button to add **density** to the **Definition** field.
- b. Click the **X** button to add the multiplication symbol to the **Definition** field.

- c. Select Velocity... and Velocity Magnitude from the Field Functions drop-down lists, and click the Select button to add |V| to the Definition field.
- d. Click **y**^**x** to raise the last entry in the **Definition** field to a power, and click **2** for the power.
- e. Click the / button to add the division symbol to the **Definition** field, and then click **2**.
- f. Enter dynamic-head for New Function Name.
- g. Click Define and close the Custom Field Function Calculator dialog box.

The **dynamic-head** tree item will appear under the **Parameters & Customization/Custom Field Functions** tree branch.

7. Display filled contours of the custom field function (Figure 3.11: Contours of the Dynamic Head Custom Field Function (p. 170)).

10001 10001 10	Postprocessing	$J \rightarrow Graphics \rightarrow$	Contours →	Edit
----------------	----------------	--------------------------------------	------------	------

Contours	
Options Filled	Contours of Custom Field Functions
Node Values	dynamic-head 👻
<ul> <li>Global Range</li> <li>Auto Range</li> </ul>	Min Max
Clip to Range	0.1365518 1.418624
Draw Profiles Draw Mesh	Surfaces Filter Text
	default-interior
Coloring	pressure-outlet-7 symmetry
<ul> <li>Banded</li> <li>Smooth</li> </ul>	velocity-inlet-5
	velocity-inlet-6 wall
Levels Setup 20 💠 1 🌩	z=0_outlet
20 • 1 •	New Surface 🔻
	Display Compute Close Help

#### a. Select Custom Field Functions... and dynamic-head from the Contours of drop-down lists.

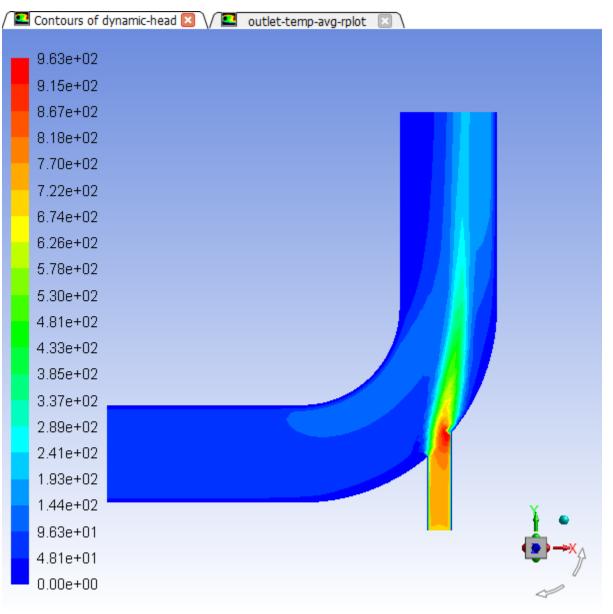
### Tip

**Custom Field Functions...** is at the top of the upper **Contours of** drop-down list.

b. Ensure that **symmetry** is selected from the **Surfaces** selection list.

c. Click **Display** and close the **Contours** dialog box.





#### Note

You may need to change the view by zooming out after the last vector display, if you have not already done so.

8. Save the settings for the custom field function by writing the case and data files (elbow1.cas.gz and elbow1.dat.gz).

**File**  $\rightarrow$  Write  $\rightarrow$  Case & Data...

a. Ensure that elbow1.cas.gz is entered for Case/Data File.

#### Note

When you write the case and data file at the same time, it does not matter whether you specify the file name with a .cas or .dat extension, as both will be saved.

- b. Click **OK** to save the files and close the **Select File** dialog box.
- c. Click **OK** to overwrite the files that you had saved earlier.

# 3.4.8. Using the Coupled Solver

The elbow solution computed in the first part of this tutorial used the SIMPLE solver scheme for Pressure-Velocity coupling. For many general fluid-flow problems, convergence speed can be improved by using the Coupled solver. You will now change the **Solution Method** to a coupled scheme and compare the convergence speed of the SIMPLE and Coupled solvers.

1. Change the solver settings.



Task Page ×
Solution Methods
Pressure-Velocity Coupling
Scheme
Coupled
Spatial Discretization
Gradient
Least Squares Cell Based 🔹
Pressure
Second Order
Momentum
Second Order Upwind
Turbulent Kinetic Energy
First Order Upwind
Turbulent Dissipation Rate
First Order Upwind
Transient Formulation
<b></b>
Non-Iterative Time Advancement
Frozen Flux Formulation
Pseudo Transient
Warped-Face Gradient Correction
High Order Term Relaxation Options
Default
Help

- a. Select **Coupled** from the **Scheme** drop-down list.
- b. Leave the **Spatial Discretization** options at their default settings.
- 2. Re-initialize the flow field.



3. Run the solution for an additional 90 iterations.

**Solving**  $\rightarrow$  Run Calculation

Run Calculation			
Update Dynamic Mesh	No. of Iterations 90		
Input Summary		-/-	
Advanced	Check Case	Calculate	

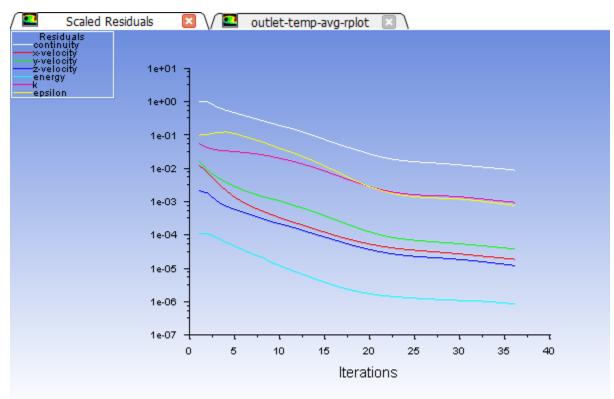
a. Enter 90 for **No. of Iterations**.

#### b. Click Calculate.

A dialog box will appear stating that outlet-temp-avg-rfile.out already exists, and asking if you want to create a new file. Click **No**. Another dialog box will appear asking whether it is OK to overwrite the file. Click **Yes**.

The solution will converge in approximately 36 iterations (Figure 3.12: Residuals for the Coupled Solver Calculation (p. 173)). Note that this is faster than the convergence rate using the SIMPLE pressure-velocity coupling. The convergence history is shown in Figure 3.13: Convergence History of Mass-Weighted Average Temperature (p. 174).

Figure 3.12: Residuals for the Coupled Solver Calculation



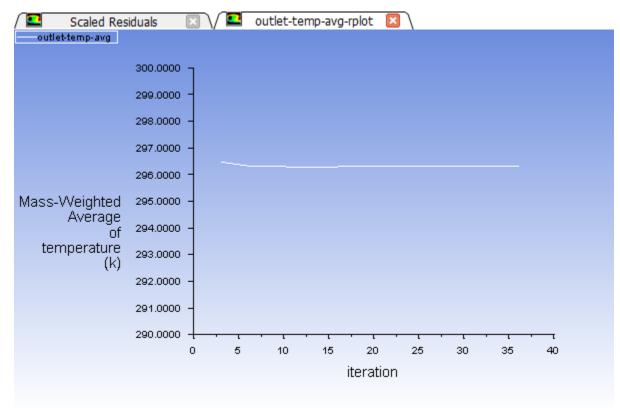


Figure 3.13: Convergence History of Mass-Weighted Average Temperature

# 3.4.9. Adapting the Mesh

For the first two runs of this tutorial, you have solved the elbow problem using a fairly coarse mesh. The elbow solution can be improved further by refining the mesh to better resolve the flow details. ANSYS Fluent provides a built-in capability to easily adapt (locally refine) the mesh according to solution gradients. In the following steps you will adapt the mesh based on the temperature gradients in the current solution and compare the results with the previous results.

1. Adapt the mesh in the regions of high temperature gradient. For this task, you will use the **Adapt** group box in the **Setting Up Domain** ribbon tab.



Setting Up Domain ightarrow Adapt ightarrow Mark/Adapt Cells ightarrow Gradient...

Cradient Adaption				<b>×</b>		
Options	Method	Gradients of				
Refine	Ourvature	Temperature		•		
Coarsen	Gradient	Static Temperature		-		
🔲 Normalize per Zone	Iso-Value	Min	Max			
Contours	Normalization	1.421085e-14	0.02847574			
Manage	Standard	Coarsen Threshold	Refine Threshold			
Controls	<ul> <li>Scale</li> <li>Normalize</li> </ul>	0	0.003			
concrossii						
	Dynamic					
	Dynamic					
	Interval					
	20 🗘					
	Adapt Mark	Compute Apply	Close Help			

a. Ensure that **Refine** is enabled in the **Options** group box.

ANSYS Fluent will not coarsen beyond the original mesh for a 3D mesh. Hence, it is not necessary to deselect **Coarsen** in this instance.

- b. Select Temperature... and Static Temperature from the Gradients of drop-down lists.
- c. Click Compute.

ANSYS Fluent will update the **Min** and **Max** values to show the minimum and maximum temperature gradient.

d. Enter 0.003 for Refine Threshold.

A general rule is to use 10% of the maximum gradient when setting the value for **Refine Threshold**.

e. Click Mark.

ANSYS Fluent will report in the console that approximately 1289 cells were marked for refinement.

f. Click Manage... to open the Manage Adaption Registers dialog box.

💶 Manage Adaptio	n Registers		
Register Actions Change Type Combine Delete Mark Actions Exchange Invert Limit Fill	Registers [1/1] gradient-r0		Register Info gradient-r0 Reg ID: 0 Refn #: 1289 Crsn #: 0 Type: adapt Options Controls
	Adapt Displ	ay Close Help	

### i. Click **Display**.

ANSYS Fluent will display the cells marked for adaption in the graphics window (Figure 3.14: Cells Marked for Adaption (p. 177)).

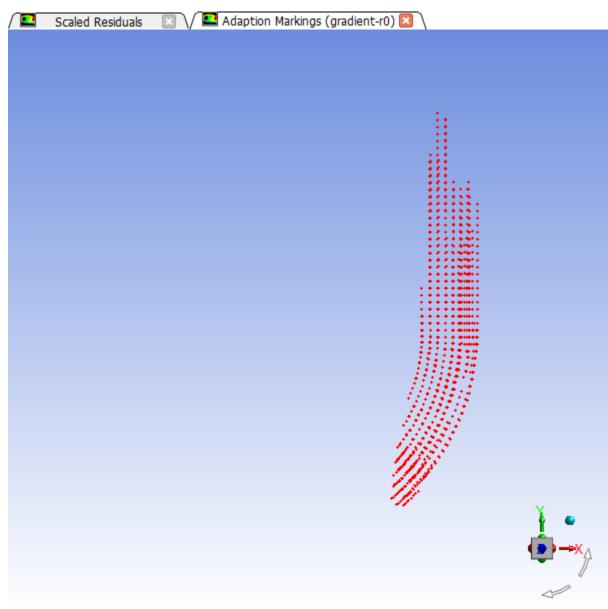


Figure 3.14: Cells Marked for Adaption

**Extra** You can change the way ANSYS Fluent displays cells marked for adaption (Figure 3.15: Alternative Display of Cells Marked for Adaption (p. 179)) by performing the following steps:

A. Click **Options...** in the **Manage Adaption Registers** dialog box to open the **Adaption Display Options** dialog box.

Adaption Display Options				
Options Draw Mesh Filled	Refine Wire Mark Color red Size 0.1	eframe	Color Cyan Size 0.1	eframe

- B. Enable **Wireframe** in the **Refine** group box.
- C. Enable **Filled** in the **Options** group box.
- D. Enable **Draw Mesh** in the **Options** group box.

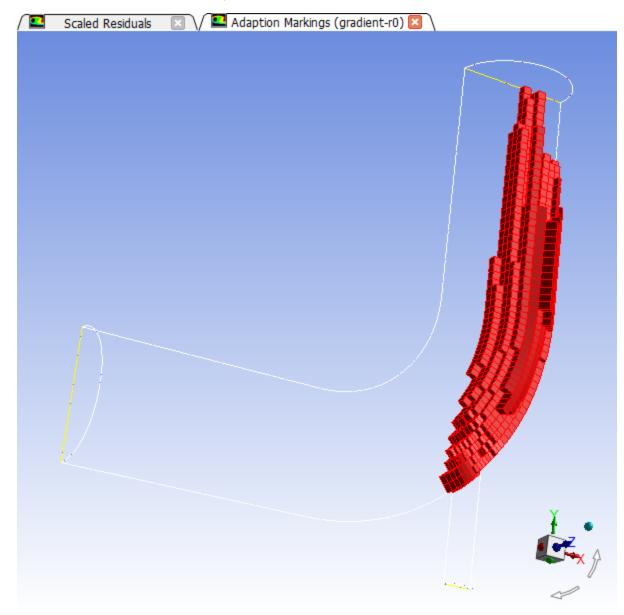
The **Mesh Display** dialog box will open.

💶 Mesh Display	,	
Options Nodes	Edge Type All	Surfaces Filter Text
Edges	Feature	default-interior
Faces	Outline	pressure-outlet-7
Partitions		symmetry
Overset		velocity-inlet-5
Christie Destant		velocity-inlet-6
	eature Angle	wall
0	20	z=0_outlet
Outline	Interior	
Adjacency		New Surface 🔻
Display Colors Close Help		

- E. Ensure that only the **Edges** option is enabled in the **Options** group box.
- F. Select Feature from the Edge Type list.
- G. Select all of the items except **default-interior** and **z=0\_outlet** from the **Surfaces** selection list.
- H. Click **Display** and close the **Mesh Display** dialog box.
- I. Click OK to close the Adaption Display Options dialog box.

- J. Click **Display** in the **Manage Adaption Registers** dialog box.
- K. Rotate the view and zoom in to get the display shown in Figure 3.15: Alternative Display of Cells Marked for Adaption (p. 179).

Figure 3.15: Alternative Display of Cells Marked for Adaption



- L. After viewing the marked cells, rotate the view back and zoom out again.
- M. Click **Options...** in the **Manage Adaption Registers** dialog box to open the **Adaption Display Options** dialog box again.
- N. In the **Adaption Display Options** dialog box, disable the **Draw Mesh** option and click **OK** to close it.
- ii. In the Manage Adaption Registers dialog box, ensure that gradient-r0 is selected from the Registers selection list.

iii. Click Adapt in the Manage Adaption Registers dialog box.

A **Question** dialog box will open, confirming your intention to adapt the mesh. Click **Yes** to proceed.

💶 Quest	ion 💌
?	Ok to change the mesh?
	Yes No

- iv. Close the Manage Adaption Registers dialog box.
- g. Close the Gradient Adaption dialog box.
- 2. Display the adapted mesh (Figure 3.16: The Adapted Mesh (p. 181)).

Setting Up	Domain $\rightarrow$ Mesh $\rightarrow$	Display
------------	---	---------

💶 Mesh Display	,	
Options Nodes	Edge Type All	Surfaces Filter Text
Edges	Feature	default-interior
Faces Partitions	Outline	pressure-outlet-7 symmetry
Overset		velocity-inlet-5
Shrink Factor F	Feature Angle	velocity-inlet-6 wall
0	20	z=0_outlet
Outline	Interior	
Adjacency		New Surface 🔻
	D	isplay Colors Close Help

- a. Select All from the Edge Type list.
- b. Deselect all of the highlighted items from the Surfaces selection list except for symmetry.

## Tip

To deselect all surfaces, click the **Deselect All Shown** button (**T**) at the top of the **Surfaces** selection list. Then select the desired surface from the **Surfaces** selection list.

c. Click **Display** and close the **Mesh Display** dialog box.

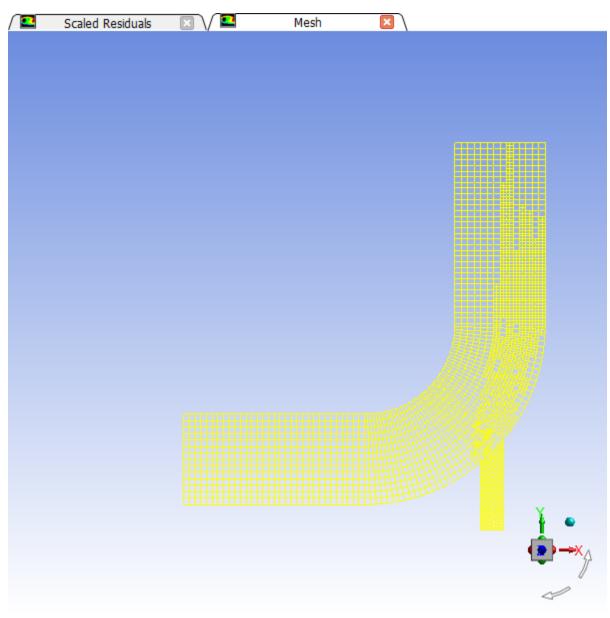
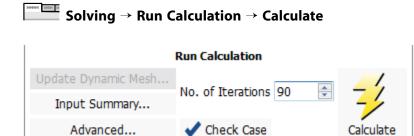


Figure 3.16: The Adapted Mesh

3. Request an additional 90 iterations.



The solution will converge after approximately 30 additional iterations (Figure 3.17: The Complete Residual History (p. 182) and Figure 3.18: Convergence History of Mass-Weighted Average Temperature (p. 182)).

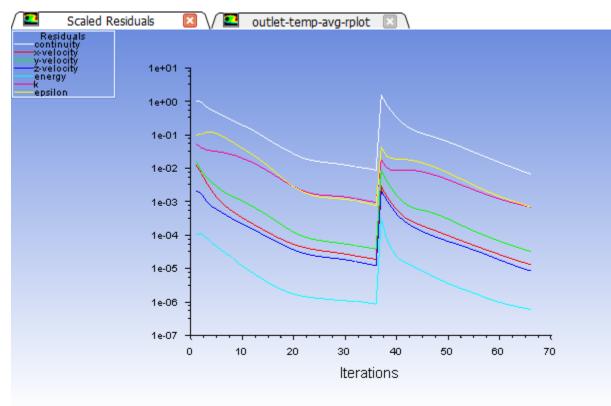
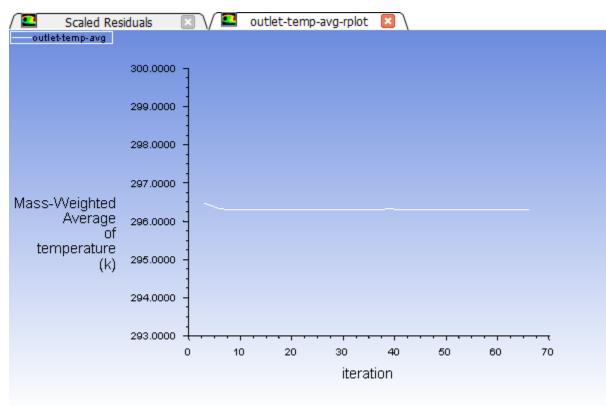


Figure 3.17: The Complete Residual History





4. Save the case and data files for the Coupled solver solution with an adapted mesh (elbow2.cas.gz and elbow2.dat.gz).

## **File** $\rightarrow$ Write $\rightarrow$ Case & Data...

- a. Enter elbow2.gz for Case/Data File.
- b. Click **OK** to save the files and close the **Select File** dialog box.

The files elbow2.cas.gz and elbow2.dat.gz will be saved in your default folder.

5. Display the filled temperature distribution (using node values) on the revised mesh using the temperature contours definition that you created earlier (Figure 3.19: Filled Contours of Temperature Using the Adapted Mesh (p. 184)).

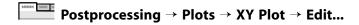
Right-click the **Results/Graphics/Contours/contour-temp** tree item and select **Display** from the menu that opens.

**Results**  $\rightarrow$  Graphics  $\rightarrow$  Contours  $\rightarrow$  contour-temp  $\stackrel{\frown}{\rightarrow}$  Display

Scaled F	Residuals 🛛 🔪 💶 Contours of St	atic Temperature (k) 🗵 🔪	
contour-temp Static Temper	rature		
3.13e			
3.12e-			
3.11e			
3.10e			
3.09e-			
3.08e-			
3.07e-			
3.06e-			
3.05e-			
3.04e			
3.03e			
3.02e			
3.01e-			
3.00e-			
2.99e-	+02		
2.98e-			
2.97e-			
2.96e			
2.95e			
2.94e			
2.93e	+02		
[k]			
			5

Figure 3.19: Filled Contours of Temperature Using the Adapted Mesh

6. Display and save an XY plot of the temperature profile across the centerline of the outlet for the adapted solution (Figure 3.20: Outlet Temperature Profile for the Adapted Coupled Solver Solution (p. 186)).



Solution XY Plot		
Options          Image: Option of the second state of the second	Plot Direction X 1 Y 0 Z 0 Load File Free Data	Y Axis Function Temperature ▼ Static Temperature X Axis Function Direction Vector Surfaces Filter Text default-interior pressure-outlet-7 symmetry velocity-inlet-5 velocity-inlet-6 wall z=0_outlet New Surface ▼
	Plot Axes Curves.	Close Help

- a. Ensure that the **Write to File** option is disabled in the **Options** group box.
- b. Ensure that **Temperature...** and **Static Temperature** are selected from the **Y Axis Function** drop-down lists.
- c. Ensure that **z=0\_outlet** is selected from the **Surfaces** selection list.
- d. Click **Plot**.

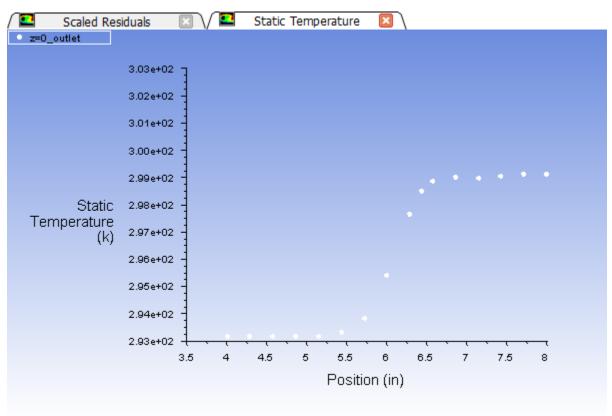


Figure 3.20: Outlet Temperature Profile for the Adapted Coupled Solver Solution

e. Enable Write to File in the Options group box.

The button that was originally labeled **Plot** will change to Write....

- f. Click Write....
  - i. In the Select File dialog box, enter outlet\_temp2.xy for XY File.
  - ii. Click **OK** to save the temperature data.
- g. Close the **Solution XY Plot** dialog box.
- 7. Display the outlet temperature profiles for both solutions on a single plot (Figure 3.21: Outlet Temperature Profiles for the Two Solutions (p. 188)).



E File XY Plot		<b>×</b>	
Plot Title Static Temperature Files	Legend Title Static Temperature Legend Entries	Add Delete Change Legend Entry	
C:/Tutorials/introduction/outlet_temp1.xy C:/Tutorials/introduction/outlet_temp2.xy	Before Adaption Adapted Mesh	( <u></u> )	
C:/Tutorials/introduction/outlet_temp2.xy Plot Axes	Adapted Mesh Curves Close Help		

a. Click the **Add...** button to open the **Select File** dialog box.

Select File	?
Look in: C:\Tutorials\introduction   C:\Tutorials\introduction  Name Solution_files Outlet_temp1.xy Outlet_temp2.xy Outlet_temp2.xy	
XY File outlet_temp2.xy	ОК
Files of type: XY Files (*.xy)	Cancel
Filter String	Filter
C:/Tutorials/introduction/outlet_temp1.xy C:/Tutorials/introduction/outlet_temp2.xy	Remove

i. Click once on **outlet\_temp1.xy** and **outlet\_temp2.xy**.

Each of these files will be listed with their folder path in the bottom list to indicate that they have been selected.

## Tip

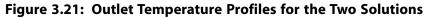
If you select a file by mistake, simply click the file in the bottom list and then click **Remove**.

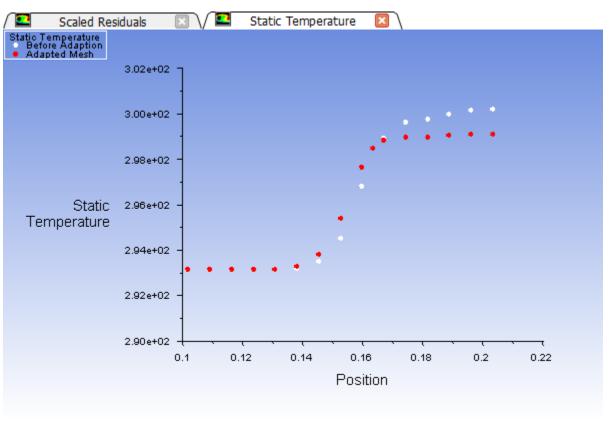
- ii. Click **OK** to save the files and close the **Select File** dialog box.
- b. Select the folder path ending in **outlet\_temp1.xy** from the **Files** selection list.
- c. Enter Before Adaption in the lower-right text-entry box.
- d. Click the Change Legend Entry button.

The item in the **Legend Entries** list for **outlet\_temp1.xy** will be changed to **Before Adaption**. This legend entry will be displayed in the upper-left corner of the XY plot generated in a later step.

- e. In a similar manner, change the legend entry for the folder path ending in **outlet\_temp2.xy** to be Adapted Mesh.
- f. Click **Plot** and close the **File XY Plot** dialog box.

Figure 3.21: Outlet Temperature Profiles for the Two Solutions (p. 188) shows the two temperature profiles at the centerline of the outlet. It is apparent by comparing both the shape of the profiles and the predicted outer wall temperature that the solution is highly dependent on the mesh and solution options. Specifically, further mesh adaption should be used in order to obtain a solution that is independent of the mesh.





#### Note

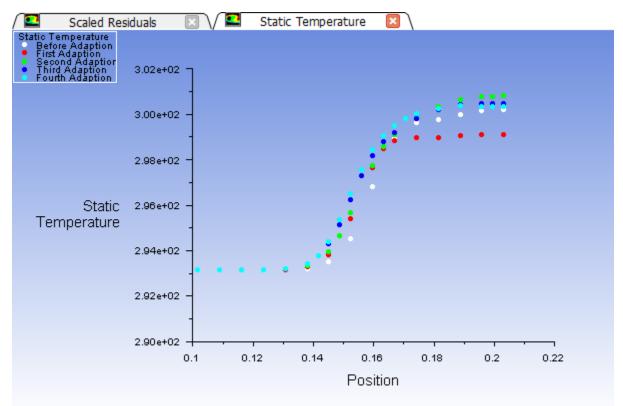
When reading and writing data values, Fluent always uses SI units. Therefore, when you read in the XY data files and plot them, the position and temperature values will be

plotted in SI units, regardless of the settings made in the **Scale Mesh** dialog box earlier in the tutorial.

#### Extra

You can perform additional rounds of mesh adaption based on temperature gradient and run the calculation to see how the temperature profile changes at the outlet. For demonstration purposes, three additional levels of mesh adaption were performed (each time using a refinement of 10% of the new maximum gradient) and the resulting temperature profiles have been plotted with outlet\_temp1.xy and outlet\_temp2.xy in Figure 3.22: Outlet Temperature Profiles for Subsequent Mesh Adaption Steps (p. 189).

Figure 3.22: Outlet Temperature Profiles for Subsequent Mesh Adaption Steps



It is evident from Figure 3.22: Outlet Temperature Profiles for Subsequent Mesh Adaption Steps (p. 189) that as the mesh is adapted further, the profiles converge on a mesh-independent profile. The resulting wall temperature at the outlet is predicted to be 300.4 *K* after mesh independence is achieved.

If computational resources allow, it is always recommended to perform successive rounds of adaption until the solution is independent of the mesh (within an acceptable tolerance). Typically, profiles of important variables are examined (in this case, temperature) and compared to determine mesh independence.

# 3.5. Setup and Solution in Parallel

This section explores the parallel processing capabilities of ANSYS Fluent. In order to be run in parallel, the mesh must be divided into smaller, evenly sized partitions. Each ANSYS Fluent process, called a compute node, will solve on a single partition, and information will be passed back and forth across all partition interfaces. The solver of ANSYS Fluent enables parallel processing on a dedicated parallel machine, or a network of workstations running Windows or Linux.

# **3.5.1. Starting the Parallel Version of ANSYS Fluent**

Since the procedure for starting the parallel version of ANSYS Fluent is dependent upon the type of machine(s) you are using, two versions of this step are provided here.

## 3.5.1.1. Multiprocessor Machine

Use ANSYS Fluent Launcher to start the 3D parallel version of ANSYS Fluent on a Windows or Linux machine using 2 processes.

- 1. Select **3D** under **Dimension**.
- 2. Enable Display Mesh After Reading under Display Options.
- 3. Make sure that **Double Precision** is disabled under **Options**.
- 4. Select Parallel under Processing Options.
- 5. Set **Processes** to 2.

To show details of the parallel settings, click **Show More Options**, then go to the **Parallel Settings** tab. Note that your **Run Types** will be **Shared Memory on Local Machine**.

6. Click **OK**.

Fluent Launcher	
<b>ANSYS</b>	Fluent Launcher
Dimension 2D 3D Display Options Display Mesh After Reading Workbench Color Scheme ACT Option Load ACT	Options Double Precision Meshing Mode Use Job Scheduler Use Remote Linux Nodes Processing Options Serial Processes 2 2 GPGPUs per Machine None
Show Fewer Options	
General Options Parallel Settings	Scheduler Environment
Interconnects default	Validate IBM MPI Password
MPI Types default Run Types Shared Memory on Local Machin Distributed Memory on a Cluster	e
OK Defa	ault Cancel Help 🔻

To start ANSYS Fluent on a Linux machine, type at the command prompt

fluent 3d -t2

If you type fluent at the command prompt, then Fluent Launcher will appear.

For additional information about parallel command line options, see the Fluent User's Guide, available in the help viewer or on the ANSYS Customer Portal (http://support.ansys.com/documentation).

## 3.5.1.2. Network of Computers

You can start the 3D parallel version of ANSYS Fluent on a network of Windows or Linux machines using 2 processes and check the network connectivity by performing the following steps:

- 1. In Fluent Launcher, restore the default settings by clicking the **Default** button.
- 2. Select **3D** under **Dimension**.
- 3. Enable Display Mesh After Reading under Display Options.
- 4. Make sure that **Double Precision** is disabled under **Options**.
- 5. Select Parallel under Processing Options.
- 6. Set the **Processes** to 2.
- 7. Click the **Show More Options** button and open the **Parallel Settings** tab.
  - a. Retain the selection of default in the Interconnects and MPI Types drop-down lists.

#### Note

On Windows platforms, default is the only available selection for Interconnects.

b. Select Distributed Memory on a Cluster under Run Types.

#### Note

The **Fluent Root Path** (in the **General Options** tab) should be specified as a network path for running Fluent across multiple machines, unless Fluent is installed in the exact same location on all of the machines that you listed for running in parallel. You

can click 🖾 to convert your **Fluent Root Path** to a network path.

- c. Make sure that File Containing Machine Names is selected to specify the file.
- d. Type the name and location of the hosts text file in the text box below **File Containing Machine Names**, or browse and select it using the **Browsing Machine File** dialog box.

Alternatively, you can select **Machine Names** and type the names of the machines in the text box.

8. Click **OK**.

#### Note

If you are using a Windows platform, you will need to enter your password for MPI the first time you run ANSYS Fluent using **Distributed Memory on a Cluster**.

E Fluent Launcher	
<b>ANSYS</b>	Fluent Launcher
Dimension 2D 3D Display Options Display Mesh After Reading Workbench Color Scheme ACT Option Load ACT	Options Double Precision Meshing Mode Use Job Scheduler Use Remote Linux Nodes Processing Options Serial Processes 2 GPGPUs per Machine None
Show Fewer Option:	None
General Options Parallel Settings	Scheduler Environment
Interconnects default	Validate IBM MPI Password
MPI Types default	•
Run Types Shared Memory on Local Machin Machine Memory on a Cluster Machine Names File Containing Machine Nam W:\fluent180\fluent.hosts	
OK Defa	ault Cancel Help 🔻

You can also start parallel ANSYS Fluent by typing the following at the command prompt:

```
fluent 3d -t2 -cnf=fluent.hosts
```

where -cnf indicates the location of the hosts text file. The hosts file is a text file that contains a list of the computers on which you want to run the parallel job. If the hosts file is not located in the directory where you are typing the startup command, you will need to supply the full pathname to the file.

For example, the fluent.hosts file may look like the following:

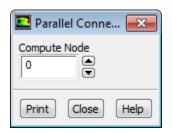
```
my_computer
another_computer
```

For additional information about hosts files and parallel command line options, see the Fluent User's Guide, available in the help viewer or on the ANSYS Customer Portal (http://support.ansys.com/ documentation).

9. Check the network connectivity information.

Although ANSYS Fluent displays a message confirming the connection to each new compute node and summarizing the host and node processes defined, you may find it useful to review the same information at some time during your session, especially if more compute nodes are spawned to several different machines.

Parallel → Network → Connectivity...



a. Set **Compute Node** to 0.

For information about all defined compute nodes, you will select node 0, since this is the node from which all other nodes are spawned.

b. Click Print.

```
ID Hostname Core O.S. PID Vendor

nl my_computer 2/32 Windows-x64 31588 Intel(R) Xeon(R) E5-2687W

n0* my_computer 1/32 Windows-x64 31736 Intel(R) Xeon(R) E5-2687W

host my_computer Windows-x64 21192 Intel(R) Xeon(R) E5-2687W

MPI Option Selected: ibmmpi

Selected system interconnect: default
```

ID is the sequential denomination of each compute node (the host process is always host), Hostname is the name of the machine hosting the compute node (or the host process), Core is the core number / total number of cores available, O.S. is the architecture, PID is the process ID number, and Vendor is the vendor of the processor. Information about the selected MPI option and system interconnect is provided at the bottom of the report.

c. Close the Parallel Connectivity dialog box.

# 3.5.2. Reading and Partitioning the Mesh

When you use the parallel solver, you need to subdivide (or partition) the mesh into groups of cells that can be solved on separate processors. If you read an unpartitioned mesh into the parallel solver, ANSYS Fluent will automatically partition it using the default partition settings. You can then check the partitions to see if you need to modify the settings and repartition the mesh.

1. Inspect the automatic partitioning settings.

# Parallel $\rightarrow$ General $\rightarrow$ Auto Partition...

Auto Partition Mesh	×
Method	Optimizations
Metis	Pre-Test
☑ Case File	
✓ Across Zones	
OK Cancel Help	

If the **Case File** option is enabled (the default setting), and there exists a valid partition section in the case file (that is, one where the number of partitions in the case file divides evenly into the number of compute nodes), then that partition information will be used rather than repartitioning the mesh. You need to disable the **Case File** option only if you want to change other parameters in the **Auto Partition Mesh** dialog box.

a. Make sure that Case File is enabled.

When the **Case File** option is enabled, ANSYS Fluent will automatically select a partitioning method for you. This is the preferred initial approach for most problems. In the next step, you will inspect the partitions created and be able to change them, if required.

- b. Click OK to close the Auto Partition Mesh dialog box.
- 2. Read the case file elbow2.cas.gz.



3. Examine the front view of the **symmetry** mesh zone (Figure 3.23: Mesh Along the Symmetry Plane for the Mixing Elbow (p. 196)).

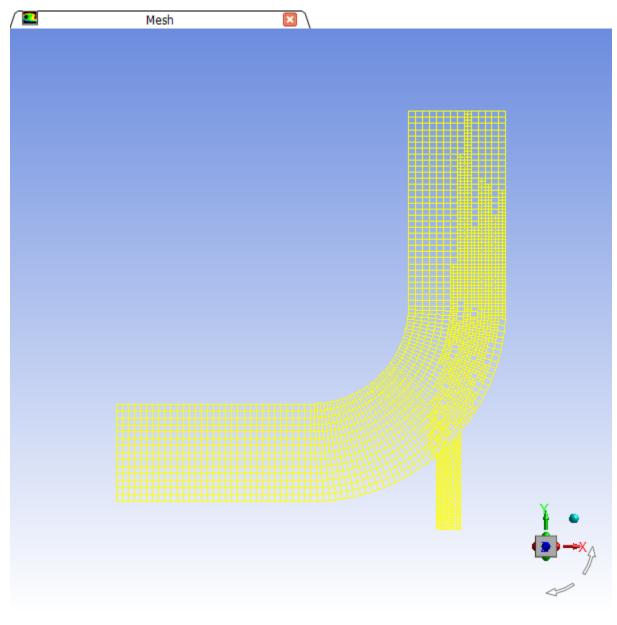


Figure 3.23: Mesh Along the Symmetry Plane for the Mixing Elbow

4. Check the partition information.

Parallel  $\rightarrow$  General  $\rightarrow$  Partition/Load Balance...

Partitioning and Load Balancing	<b>—</b>
Method	_
Metis	•
Options Optimization Weighting Dynamic Load Balancing	Zones Filter Text 🗾 📆 📆
Number of Partitions 2	fluid
Reporting Verbosity 1 🚔	
☑ Across Zones	
Laplace Smoothing	
Reordering Methods	Registers [0/0]
Architecture Aware     Reverse Cuthil-McKee	
C Reverse coulimmerce	
Print Active Partitions Print Stored Partitions Use Stored Partition	s
Set Selected Zones and Registers to Partition ID 0	
Partition Reorder Default	Close Help

### a. Click Print Active Partitions.

ANSYS Fluent will print the active partition statistics in the console.

2	Active	e Parti	tions:										
	P	Cells	I-Cells	Cell H	Ratio	Faces	I-Faces	Face	Ratio	Neighbors	Load	Ext	Cells
	0	11570	439	(	0.038	37868	633		0.017	1	1		410
	1	11305	410	(	0.036	37856	633		0.017	1	1		439
	Colle	ective	Partitic	on Stat						n Total			
	Cell	count					11305			22875			
	Mean	cell c	count dev	viation	n		-1.2%	-	1.2%				
	Part	ition k	oundary	cell d	count		410	4	139	849			
	Part	ition k	oundary	cell d	count :	ratio	3.6%		3.8%	3.7%			
	Face	count					37856		37868	75091			
	Mean	face o	count dev	viation	n		-0.0%						
	Part	ition k	oundary	face o	count		633	6	533	633			
			-			ratio	1.7%	-	1.7%	0.8%			
	Part	ition r	neighbor	count			1	-	L				
	Part	ition M	lethod				Metis						
	Store	ed Part	ition Co	ount			2						

#### Note

ANSYS Fluent distinguishes between two cell partition schemes within a parallel problem: the active cell partition, and the stored cell partition. Here, both are set to the cell partition that was created upon reading the case file. If you repartition the mesh using the **Partitioning and Load Balancing** dialog box, the new partition will be referred to as the stored cell partition. To make it the active cell partition, you need to click the **Use Stored Partitions** button in the **Partitioning and Load Balan**.

**cing** dialog box. The active cell partition is used for the current calculation, while the stored cell partition (the last partition performed) is used when you save a case file. This distinction is made mainly to allow you to partition a case on one machine or network of machines and solve it on a different one.

For details, see the chapter on parallel processing in the Fluent User's Guide, available in the help viewer or on the ANSYS Customer Portal (http://support.ansys.com/documentation).

b. Review the partition statistics.

An optimal partition should produce an equal number of cells in each partition for load balancing, a minimum number of partition interfaces to reduce interpartition communication, and a minimum number of partition neighbors to reduce the startup time for communication. In the displayed partition statistics, you will be looking for relatively small values of mean cell and face count deviation, and total partition boundary cell and face count ratio. Values less than 5% are considered reasonable in most cases. However, with very large core counts and/or especially complex cases, larger values may be unavoidable.

- c. Close the Partitioning and Load Balancing dialog box.
- 5. Examine the partitions graphically.
  - a. Initialize the solution using the default values.

## **Solving** $\rightarrow$ Initialization $\rightarrow$ Initialize

In order to use the **Contours** dialog box to inspect the partition you just created, you have to initialize the solution, even though you are not going to solve the problem at this point. The default values are sufficient for this initialization.

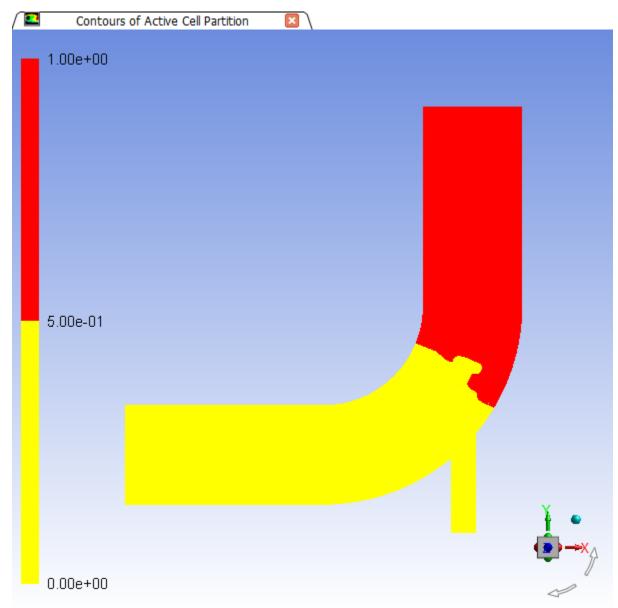
b. Display the cell partitions (Figure 3.24: Cell Partitions (p. 200)).

Postprocessing  $\rightarrow$  Graphics  $\rightarrow$  Contours  $\rightarrow$  Edit...

Contours		×			
Options Filled	Contours of Cell Info	•			
Node Values Global Range	Active Cell Partition	•			
<ul> <li>Global Range</li> <li>Auto Range</li> </ul>	Min	Max			
Clip to Range	0				
Draw Profiles Draw Mesh	Surfaces Filter Text				
Colorina	default-interior pressure-outlet-7				
Coloring Banded Smooth	symmetry velocity-inlet-5				
Levels Setup z=0 outlet					
2 😨 1 😨	2 1 New Surface				
	Display Compute	e Close Help			

- i. Make sure that **Filled** is selected in the **Options** group box.
- ii. Select Cell Info... and Active Cell Partition from the Contours of drop-down lists.
- iii. Select **symmetry** from the **Surfaces** selection list.
- iv. Set Levels to 2, which is the number of compute nodes.
- v. Click **Display** and close the **Contours** dialog box.

Figure 3.24: Cell Partitions



As shown in Figure 3.24: Cell Partitions (p. 200), the cell partitions are acceptable for this problem. The position of the interface reveals that the criteria mentioned earlier will be matched. If you are dissatisfied with the partitions, you can use the **Partitioning and Load Balancing** dialog box to repartition the mesh. Recall that, if you want to use the modified partitions for a calculation, you will need to make the stored cell partition the active cell partition by either clicking the **Use Stored Partitions** button in the **Partitioning and Load Balancing** dialog box or saving the case file and reading it back into ANSYS Fluent.

For details about the procedure and options for manually partitioning a mesh, see the Fluent User's Guide, available in the help viewer or on the ANSYS Customer Portal (http://support.an-sys.com/documentation).

6. Save the case file with the partitioned mesh (elbow4.cas.gz).

File  $\rightarrow$  Write  $\rightarrow$  Case...

## 3.5.3. Solution

1. Initialize the flow field.

Solving $\rightarrow$ Initialization					
	Initializa	ation			
Method		Patch			
O Hybrid	More Settings	Reset Statistics			
Standard	Standard Options		t = 0 Initialize		

- a. Make sure that Hybrid is selected under Method (this is the default).
- b. Click Initialize.

A **Question** dialog box will open, asking if you want to discard the data generated during the first initialization, which was used to inspect the cell partitions.

- c. Click **OK** in the **Question** dialog box to discard the data.
- 2. Enable the plotting of residuals during the calculation.



3. Start the calculation by requesting 90 iterations.

### Solving → Run Calculation

A **Question** dialog box will open, warning that outlet-temp-avg-rfile.out already exists and asking if you want to create a new file. Click **Yes**.

The solution will converge in approximately 36 iterations.

4. Save the data file (elbow4.dat.gz).

**File**  $\rightarrow$  Write  $\rightarrow$  Data...

## 3.5.4. Checking Parallel Performance

Generally, you will use the parallel solver for large, computationally intensive problems, and you will want to check the parallel performance to determine if any optimization is required. Although the example in this tutorial is a simple 3D case, you will check the parallel performance as an exercise.

For details, see the chapter on parallel processing in the Fluent User's Guide, available in the help viewer or on the ANSYS Customer Portal (http://support.ansys.com/documentation).

## Parallel $\rightarrow$ Timer $\rightarrow$ Usage

Performance Timer for 36 iterations on 2 comput	e nodes
Average wall-clock time per iteration:	0.097 sec
Global reductions per iteration:	55 ops
Global reductions time per iteration:	0.000 sec (0.0%)
Message count per iteration:	637 messages

Data transfer per iteration:	0.550 MB
LE solves per iteration:	4 solves
LE wall-clock time per iteration:	0.046 sec (47.7%)
LE global solves per iteration:	4 solves
LE global wall-clock time per iteration:	0.000 sec (0.1%)
LE global matrix maximum size:	30
AMG cycles per iteration:	6.889 cycles
Relaxation sweeps per iteration:	655 sweeps
Relaxation exchanges per iteration:	0 exchanges
Total wall-clock time:	3.499 sec

The most accurate way to evaluate parallel performance is by running the same parallel problem on 1 CPU and on *n* CPUs, and comparing the Total wall-clock time (elapsed time for the iterations) in both cases. Ideally you would want to have the Total wall-clock time with *n* CPUs be 1/n times the Total wall-clock time with 1 CPU. In practice, this improvement will be reduced by the performance of the communication subsystem of your hardware, and the overhead of the parallel process itself.

#### Note

The wall clock time and the ratio of iterations to convergence time may differ depending on the type of computer you are running (for example, Windows 64, Linux 64, and so on).

### 3.5.5. Postprocessing

Here, two plots are generated so that you can confirm that the results obtained with the parallel solver are the same as those obtained previously with the serial solver.

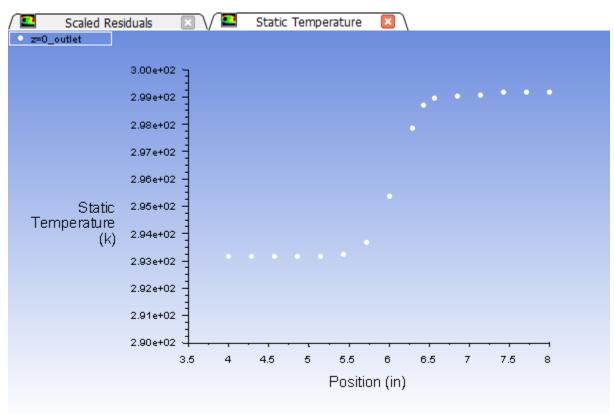
1. Display an XY plot of temperature along the centerline of the outlet (Figure 3.25: Temperature Distribution at the Outlet (p. 203)).

Solution XY Plot			<b>×</b>	
Options Vode Values V Position on X Axis Position on Y Axis Vrite to File Order Points File Data		Plot Direction X 1 Y 0 Z 0	Y Axis Function Temperature Static Temperature X Axis Function Direction Vector Surfaces Fiter Text default-interior pressure-outlet-7 symmetry velocity-inlet-5 velocity-inlet-5 velocity-inlet-6 wall z=0_outlet New Surface New Surface	
Plot Axes Curves Close Help				

#### Postprocessing $\rightarrow$ Plots $\rightarrow$ XY Plot $\rightarrow$ Edit...

- a. Select Temperature... and Static Temperature from the Y Axis Function drop-down lists.
- b. Select **z=0\_outlet** from the **Surfaces** selection list.
- c. Click Plot and close the Solution XY Plot dialog box.

Figure 3.25: Temperature Distribution at the Outlet



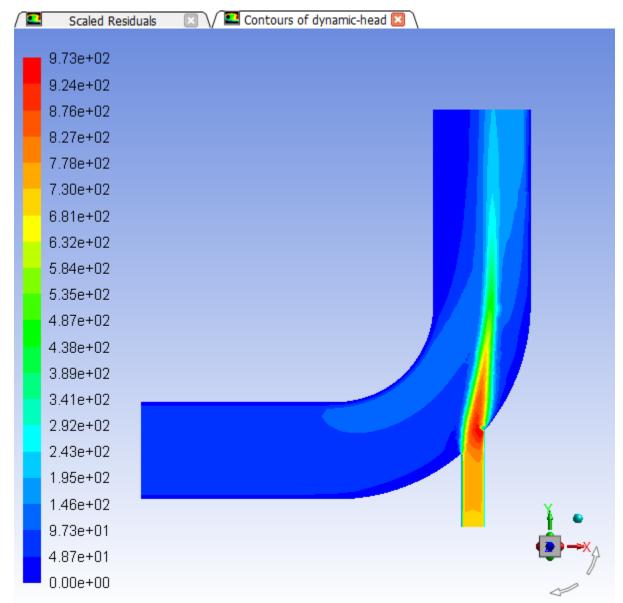
Compare the plot of Temperature at the Outlet with the serial solution shown in Figure 3.20: Outlet Temperature Profile for the Adapted Coupled Solver Solution (p. 186).

2. Display filled contours of the custom field function **dynamic-head** (Figure 3.26: Contours of the Custom Field Function, Dynamic Head (p. 205)).



Contours	
Options Filled Options Opti	Contours of Custom Field Functions
	dynamic-head 🔹
Auto Range Clip to Range	0 973.0163
Draw Profiles Draw Mesh	Surfaces Filter Text
Coloring	default-interior pressure-outlet-7
<ul> <li>Banded</li> <li>Smooth</li> </ul>	symmetry velocity-inlet-5 velocity-inlet-6
Levels Setup	wall z=0_outlet
20 • 1 •	New Surface 🔻
	Display Compute Close Help

- a. Select **Custom Field Functions...** and **dynamic-head** from the **Contours of** drop-down lists.
- b. Enter 20 for Levels.
- c. Select **symmetry** from the **Surfaces** selection list.
- d. Click **Display** and close the **Contours** dialog box.



#### Figure 3.26: Contours of the Custom Field Function, Dynamic Head

## 3.6. Summary

A comparison of the convergence speed for the SIMPLE and Coupled pressure-velocity coupling schemes indicates that the latter converges much faster. With more complex meshes, the difference in speed between the two schemes can be significant.

# **Chapter 4: Modeling Periodic Flow and Heat Transfer**

This tutorial is divided into the following sections:

- 4.1. Introduction
- 4.2. Prerequisites
- 4.3. Problem Description
- 4.4. Setup and Solution
- 4.5. Summary
- 4.6. Further Improvements

## 4.1. Introduction

Many industrial applications, such as steam generation in a boiler or air cooling in the coil of an air conditioner, can be modeled as two-dimensional periodic heat flow. This tutorial illustrates how to set up and solve a periodic heat transfer problem, given a pre-generated mesh.

The system that is modeled is a bank of tubes containing a flowing fluid at one temperature that is immersed in a second fluid in cross flow at a different temperature. Both fluids are water, and the flow is classified as laminar and steady, with a Reynolds number of approximately 100. The mass flow rate of the cross flow is known and the model is used to predict the flow and temperature fields that result from convective heat transfer.

Due to symmetry of the tube bank and the periodicity of the flow inherent in the tube bank geometry, only a portion of the geometry will be modeled in ANSYS Fluent, with symmetry applied to the outer boundaries. The resulting mesh consists of a periodic module with symmetry. In the tutorial, the inlet boundary will be redefined as a periodic zone, and the outflow boundary defined as its shadow.

This tutorial demonstrates how to do the following:

- Create periodic zones.
- Define a specified periodic mass flow rate.
- Model periodic heat transfer with specified temperature boundary conditions.
- Calculate a solution using the pressure-based, pseudo-transient, coupled solver.
- Plot temperature profiles on specified isosurfaces.

## 4.2. Prerequisites

This tutorial is written with the assumption that you have completed one or more of the introductory tutorials found in this manual:

- Introduction to Using ANSYS Fluent in ANSYS Workbench: Fluid Flow and Heat Transfer in a Mixing Elbow (p. 1)
- Parametric Analysis in ANSYS Workbench Using ANSYS Fluent (p. 73)

• Introduction to Using ANSYS Fluent: Fluid Flow and Heat Transfer in a Mixing Elbow (p. 121)

and that you are familiar with the ANSYS Fluent tree and ribbon structure. Some steps in the setup and solution procedure will not be shown explicitly.

## 4.3. Problem Description

This problem considers a 2D section of a tube bank. A schematic of the problem is shown in Figure 4.1: Schematic of the Problem (p. 208). The bank consists of uniformly-spaced tubes with a diameter of 1 cm, which are staggered across the cross-fluid flow. Their centers are separated by a distance of 2 cm in the x direction, and 1 cm in the y direction. The bank has a depth of 1 m.

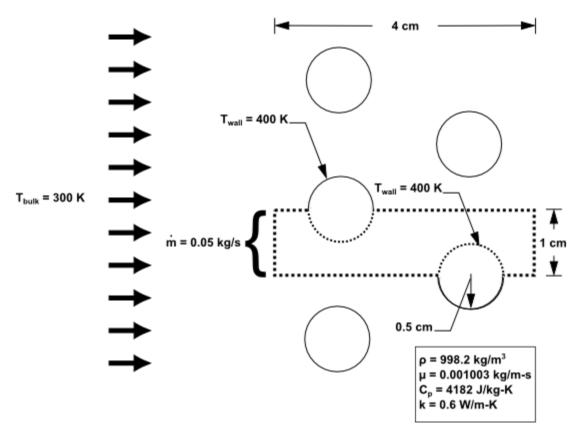


Figure 4.1: Schematic of the Problem

Because of the symmetry of the tube bank geometry, only a portion of the domain must be modeled. The computational domain is shown in outline in Figure 4.1: Schematic of the Problem (p. 208). A mass flow rate of 0.05 kg/s is applied to the inlet boundary of the periodic module. The temperature of the tube wall ( $T_{wall}$ ) is 400 K and the bulk temperature of the cross flow water ( $T_{bulk}$ ) is 300 K. The properties of water that are used in the model are shown in Figure 4.1: Schematic of the Problem (p. 208).

## 4.4. Setup and Solution

The following sections describe the setup and solution steps for this tutorial:

4.4.1. Preparation4.4.2. Mesh4.4.3. General Settings4.4.4. Models4.4.5. Materials

4.4.6. Cell Zone Conditions4.4.7. Periodic Conditions4.4.8. Boundary Conditions4.4.9. Solution4.4.10. Postprocessing

### 4.4.1. Preparation

To prepare for running this tutorial:

- 1. Set up a working folder on the computer you will be using.
- 2. Go to the ANSYS Customer Portal, https://support.ansys.com/training.

#### Note

If you do not have a login, you can request one by clicking **Customer Registration** on the log in page.

- 3. Enter the name of this tutorial into the search bar.
- 4. Narrow the results by using the filter on the left side of the page.
  - a. Click ANSYS Fluent under Product.
  - b. Click 18.0 under Version.
- 5. Select this tutorial from the list.
- 6. Click the **periodic\_flow\_heat\_R180.zip** link to download the input files.
- 7. Unzip periodic\_flow\_heat\_R180.zip to your working folder.

The file tubebank.msh can be found in the periodic\_flow\_heat folder created after unzipping the file.

8. Use Fluent Launcher to start the **2D** version of ANSYS Fluent.

Fluent Launcher displays your **Display Options** preferences from the previous session.

For more information about the Fluent Launcher, see starting ANSYS Fluent using the Fluent Launcher in the Fluent Getting Started Guide.

- 9. Ensure that the **Display Mesh After Reading** and **Workbench Color Scheme** options are enabled.
- 10. Ensure that you are running in single precision (disable **Double Precision**).
- 11. Select Serial under Processing Options.

### 4.4.2. Mesh

1. Read the mesh file tubebank.msh.

File  $\rightarrow$  Read  $\rightarrow$  Mesh...

2. Check the mesh.



ANSYS Fluent will perform various checks on the mesh and report the progress in the ANSYS Fluent console window. Ensure that the minimum volume reported is a positive number.

3. Scale the mesh.

### Setting Up Domain $\rightarrow$ Mesh $\rightarrow$ Scale...

💶 Scale M	esh			<b>—</b>	
- Domain E	Extents			Scaling	
Xmin (m)	0	Xmax (m)	0.04	Onvert Units	
Ymin (m)	0	Ymax (m)	0.01	Specify Scaling Factors	
				Mesh Was Created In	
				cm 🔻	
View Leng	th Unit In			Scaling Factors	
m	•			X 0.01	
				Y 0.01	
Scale Unscale					
Close Help					

- a. Select **cm** (centimeters) from the **Mesh Was Created In** drop-down list in the **Scaling** group box.
- b. Click Scale to scale the mesh.
- c. Close the Scale Mesh dialog box.
- 4. Check the mesh.

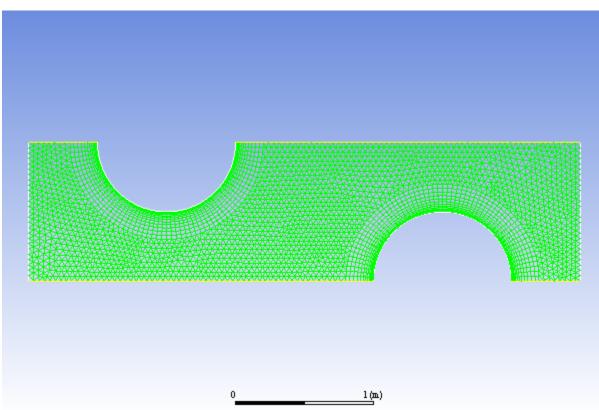
Setting Up Domain  $\rightarrow$  Mesh  $\rightarrow$  Check

#### Note

It is a good idea to check the mesh after you manipulate it (scale, convert to polyhedra, merge, separate, fuse, add zones, or smooth and swap.) This will ensure that the quality of the mesh has not been compromised.

5. Examine the mesh (Figure 4.2: Mesh for the Periodic Tube Bank (p. 211)).





Quadrilateral cells are used in the regions surrounding the tube walls and triangular cells are used for the rest of the domain, resulting in a hybrid mesh (see Figure 4.2: Mesh for the Periodic Tube Bank (p. 211)). The quadrilateral cells provide better resolution of the viscous gradients near the tube walls. The remainder of the computational domain is filled with triangular cells for the sake of convenience.

#### Tip

You can use the right mouse button to probe for mesh information in the graphics window. If you click the right mouse button on any node in the mesh, information will be displayed in the ANSYS Fluent console about the associated zone, including the name of the zone. This feature is especially useful when you have several zones of the same type and you want to distinguish between them quickly.

6. Create the periodic zone.

The inlet (**wall-9**) and outflow (**wall-12**) boundaries currently defined as wall zones need to be redefined as periodic using the text user interface. The **wall-9** boundary will be redefined as a translationally periodic zone and **wall-12** as a periodic shadow of **wall-9**.

- a. Press Enter in the console to get the command prompt (>).
- b. Enter the text command and input the responses to the prompts as shown. Press Enter after each entry.

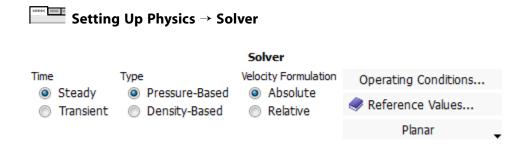
> mesh/modify-zones/make-periodic Enter

You will be prompted to enter the zones corresponding to the periodic boundaries and specify the configuration of the periodicity.

Periodic zone [()] 9 Enter
Shadow zone [()] 12 Enter
Rotational periodic? (if no, translational) [yes] no Enter
Create periodic zones? [yes] yes Enter
Auto detect translation vector? [yes] yes Enter
zone 12 deleted
created periodic zones.

## 4.4.3. General Settings

1. Retain the default settings for the solver.



### 4.4.4. Models

1. Enable heat transfer.

Setting Up Physics  $\rightarrow$  Models  $\rightarrow$  Energy

#### 4.4.5. Materials

The default properties for water defined in ANSYS Fluent are suitable for this problem. In this step, you will make sure that this material is available for selecting in future steps.

1. Add water to the list of fluid materials by copying it from the ANSYS Fluent materials database.



a. Click **Fluent Database...** in the **Create/Edit Materials** dialog box to open the **Fluent Database Materials** dialog box.

Fluent Database Materials		<b>X</b>
Fluent Fluid Materials [1/563]		Material Type
vinyl-silylidene (h2cchsih) vinyl-trichlorosilane (sicl3ch2ch) vinylidene-chloride (ch2ccl2)	*	Order Materials by Name  Chemical Formula
water-liquid (h2o <l>) water-vapor (h2o)</l>		
wood-volatiles (wood_vol) Copy Materials from Case Delete	•	
Properties		
Density (kg/m3)	constant	▼ View ▲
	998.2	
Cp (Specific Heat) (j/kg-k)	constant	▼ View
	4182	
Thermal Conductivity (w/m-k)	constant	▼ View
	0.6	
Viscosity (kg/m-s)	constant	View
	0.001003	
New Edit	. Save Copy Clo	se Help

i. Select water-liquid (h2o<l>) in the Fluent Fluid Materials selection list.

Scroll down the list to find **water-liquid** (**h2o**<**I**>**)**. Selecting this item will display the default properties in the dialog box.

ii. Click Copy and close the Fluent Database Materials dialog box.

The **Create/Edit Materials** dialog box will now display the copied properties for water-liquid.

Name	Material Type	Order Materials by
water-liquid	fluid	
Chemical Formula	Fluent Fluid Materials	Chemical Formula
h2o <l></l>	water-liquid (h2o <l>) Mixture none</l>	Fluent Database
Properties		_
Density (kg/m3) constant 998.2 Cp (Specific Heat) (j/kg-k) constant 4182 Thermal Conductivity (w/m-k) constant 0.6 Viscosity (kg/m-s) constant	<ul> <li>▼ Edit</li> <li>▼ Edit</li> <li>■</li> <li>■</li></ul>	
	Change/Create Delete Close Help	

b. Click Change/Create and close the Create/Edit Materials dialog box.

## 4.4.6. Cell Zone Conditions

1. Set the cell zone conditions for the continuum fluid zone (**fluid-16**).

<b>E</b> Setup $\rightarrow$ <b>Cell Zone Conditions</b> $\rightarrow \stackrel{=}{=} $ fluid-16 $\rightarrow$ Edit.
--

E Fluid								
Zone Name				_				
fluid-16								
Material Name water	r-liquid	▼ Edit						
Frame Motion	Source Terms							
Mesh Motion	Fixed Values							
Porous Zone								
Reference Frame	Mesh Motion	Porous Zone	3D Fan Zone	Embedded LES	Reaction	Source Terms	Fixed Values	Multiphase
Rotation-Axis Ori	gin							
X (m) 0	constant	•						
Y (m) 0	constant	•						
OK Cancel Heb								

- a. Select water-liquid from the Material Name drop-down list.
- b. Click **OK** to close the **Fluid** dialog box.

## 4.4.7. Periodic Conditions

1. Define the periodic flow conditions.

Setup →	$ \mathbf{O} \mathbf{Boundary} \mathbf{Conditions} \rightarrow \mathbf{E} \mathbf{E} \mathbf{Conditions} \mathbf{O} \mathbf{Conditions} \mathbf{O} \mathbf{E} \mathbf{Conditions} \mathbf{O} \mathbf{Conditions} \mathbf{O} \mathbf{E} \mathbf{Conditions} \mathbf{O} \mathbf{E} \mathbf{Conditions} \mathbf{O} \mathbf{E} \mathbf{Conditions} \mathbf{O} Co$	periodic-9 → Periodic Conditions
---------	--	----------------------------------

Periodic Conditions	<b>—</b>
<ul> <li>Type</li> <li>Specify Mass Flow</li> <li>Specify Pressure Gradient</li> </ul>	Flow Direction X 1 Y 0 Z 0
Mass Flow Rate (kg/s)	Relaxation Factor
0.05	0.5
Pressure Gradient (pascal/m)	Number of Iterations
0	2
Upstream Bulk Temperature (k)	
300	
OK Update	Cancel Help

a. Select Specify Mass Flow in the Type list.

This will allow you to specify the Mass Flow Rate.

- b. Enter 0.05 kg/s for Mass Flow Rate.
- c. Click OK to close the Periodic Conditions dialog box.

### 4.4.8. Boundary Conditions

1. Set the boundary conditions for the bottom wall of the left tube (wall-21).

```
E Setup \rightarrow Conditions \rightarrow E wall-21 \rightarrow Edit...
```

💶 Wall		×
Zone Name		
wall-bottom		
Adjacent Cell Zone		
fluid-16		
Momentum Thermal	Radiation Species DPM Multiphase UDS Wall Film Potential	_
Thermal Conditions		
Heat Flux	Temperature (k) 400 constant	-
Temperature	Wall Thickness (m) 0	P
Convection		-
Radiation		
Mixed		
<ul> <li>via System Coupling</li> <li>via Mapped Interface</li> </ul>		
Material Name		
aluminum 🔻	Edit	
		=
	OK Cancel Help	

- a. Enter wall-bottom for Zone Name.
- b. Click the **Thermal** tab.
  - i. Select Temperature in the Thermal Conditions list.
  - ii. Enter 400 K for Temperature.

These settings will specify a constant wall temperature of 400 K.

- c. Click **OK** to close the **Wall** dialog box.
- 2. Set the boundary conditions for the top wall of the right tube (**wall-3**).

Setup  $\rightarrow$   $\bigcirc$  Boundary Conditions  $\rightarrow \stackrel{\frown}{=}$  wall-3  $\rightarrow$  Edit...

- a. Enter wall-top for **Zone Name**.
- b. Click the **Thermal** tab.
  - i. Select Temperature from the Thermal Conditions list.
  - ii. Enter 400 K for **Temperature**.
- c. Click **OK** to close the **Wall** dialog box.

#### 4.4.9. Solution

1. Set the solution parameters.

Solving  $\rightarrow$  Solution  $\rightarrow$  Methods...

Solution Methods
Pressure-Velocity Coupling
Scheme
Coupled
Spatial Discretization
Gradient
Least Squares Cell Based 🔹
Pressure
Second Order 🔹
Momentum
Second Order Upwind 🔹
Energy
Second Order Upwind 🔹
Transient Formulation
<b></b>
Non-Iterative Time Advancement
Frozen Flux Formulation
Pseudo Transient
Warped-Face Gradient Correction
High Order Term Relaxation Options
Default

- a. Select Coupled from the Scheme drop-down list in the Pressure-Velocity Coupling group box.
- b. Retain the default setting of **Least Squares Cell Based** for the **Gradient** in the **Spatial Discretization** group box.
- c. Retain the default setting of Second Order for the Pressure drop-down list.
- d. Retain the default setting of Second Order Upwind in the Momentum and Energy drop-down lists.
- e. Enable **Pseudo Transient**.

The Pseudo Transient option enables the pseudo transient algorithm in the coupled pressure-based solver. This algorithm effectively adds an unsteady term to the solution equations in order to improve stability and convergence behavior.

2. Set the solution controls.

**Solving**  $\rightarrow$  Controls  $\rightarrow$  Controls...

Solution Controls	
Pseudo Transient Explicit Relaxation Factors	
Pressure	-
0.5	
Momentum	
0.5	
Density	E
1	
Body Forces	
1	
Energy	
0.75	
	Ŧ
Default	
Equations Limits Advanced	

a. Retain the default values in the **Pseudo Transient Explicit Relaxation Factors** group box.

In some cases, the default Pseudo Transient Explicit Relaxation Factors may need to be reduced in order to prevent oscillation of residual values or stabilization of residual values above the convergence criteria. For additional information about setting Pseudo Transient Explicit Relaxation Factors, see the Fluent User's Guide.

3. Enable the plotting of residuals during the calculation.

Residual Monitors					×
Options	Equations				
Print to Console	Residual	Monitor	Check Converge	ence Absolute Criteria	<u> </u>
✓ Plot	continuity	V	$\mathbf{V}$	0.001	
Window	x-velocity	<b>V</b>	$\checkmark$	0.001	E
1 Curves Axes	y-velocity	<b>V</b>	<b>V</b>	0.001	
Iterations to Plot	energy		$\checkmark$	1e-06	
1000 🗢	Residual Values			Convergence Criterio	n
	📃 Normalize		Iterations	absolute	•
Iterations to Store			5 🗼		
1000 ≑	Scale			Convergence Condit	ions
	Compute Loca	l Scale			
OK Plot Renormalize Cancel Help					

#### Solving $\rightarrow$ Reports $\rightarrow$ Residuals...

- a. Ensure **Plot** is enabled in the **Options** group box.
- b. Click **OK** to close the **Residual Monitors** dialog box.

4. Initialize the solution.

You will first use the default Hybrid initialization method and then patch the fluid zone with the upstream temperature value.

a. Initialize using the default settings.

#### **Solving** $\rightarrow$ Initialization $\rightarrow$ Initialize

b. Patch the fluid zone with the bulk upstream temperature value.

The Hybrid Initialization method computes the initial flow field based on inlet and outlet boundary conditions. In this case we have periodic boundary conditions with a specified upstream bulk temperature. You will patch the initialized solution with this temperature value in order to improve convergence.

#### Solving $\rightarrow$ Initialization $\rightarrow$ Patch...

Patch			<b>×</b>
Reference Frame <ul> <li>Relative to Cell Zone</li> <li>Absolute</li> </ul> Variable           Pressure           X Velocity           Y Velocity           Temperature           Phi for wall distance	Value (k) 300 Use Field Function Field Function	Zones to Patch Filter Text  fluid-16  Registers to Patch [0/0]	
	Patch Clos	e Help	

- i. Select Temperature in the Variable selection list.
- ii. Enter 300 for Value (k).

Recall that the upstream bulk temperature,  $T_{bulk}$ , is specified as 300 K.

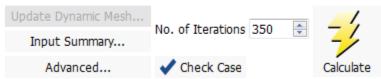
- iii. Select fluid-16 in the Zones to Patch selection list.
- iv. Click **Patch** and close the **Patch** dialog box.
- 5. Save the case file (tubebank.cas.gz).



6. Start the calculation by requesting 350 iterations.

Solving → Run Calculation

#### Run Calculation



- a. Enter 350 for No. of Iterations.
- b. Click Calculate.

The solution will converge in approximately 111 iterations.

7. Save the case and data files (tubebank.cas.gz and tubebank.dat.gz).



### 4.4.10. Postprocessing

1. Display filled contours of static pressure (Figure 4.3: Contours of Static Pressure (p. 221)).

Contours	<b>—</b>
Options Filled	Contours of Pressure
<ul> <li>Node Values</li> <li>Global Range</li> </ul>	Static Pressure 👻
Auto Range	Min Max
Clip to Range Draw Profiles	
Draw Mesh	Surfaces Filter Text
Coloring Banded Smooth Levels Setup 20 1	interior-15  periodic-9 symmetry-11 symmetry-13 symmetry-18 symmetry-24 wall-bottom
20 • 1 •	New Surface 🔻
	Display Compute Close Help

- a. Enable Filled in the Options group box.
- b. Retain the default selection of Pressure... and Static Pressure from the Contours of drop-down lists.
- c. Click Display.

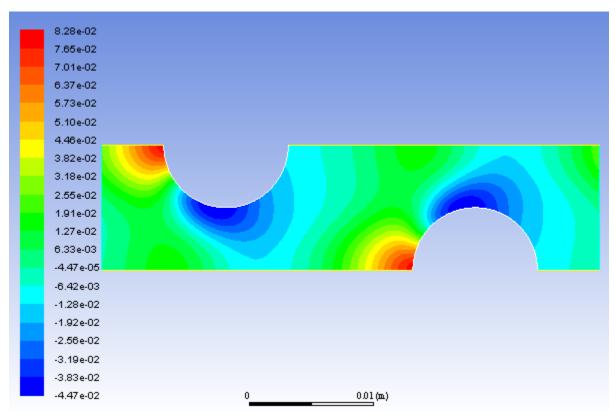


Figure 4.3: Contours of Static Pressure

d. Change the view to mirror the display across the symmetry planes (Figure 4.4: Contours of Static Pressure with Symmetry (p. 222)).

Views Views back bottom front isometric left	Actions Default Auto Scale Previous Save	Mirror Planes [4/4]
left right top Save Name view-0	Delete Read Write	Symmetry-24 Define Plane Periodic Repeats Define Close Help

 $\blacksquare Viewing \rightarrow Display \rightarrow Views...$ 

i. Select all of the symmetry zones (symmetry-18, symmetry-13, symmetry-11, and symmetry-24)

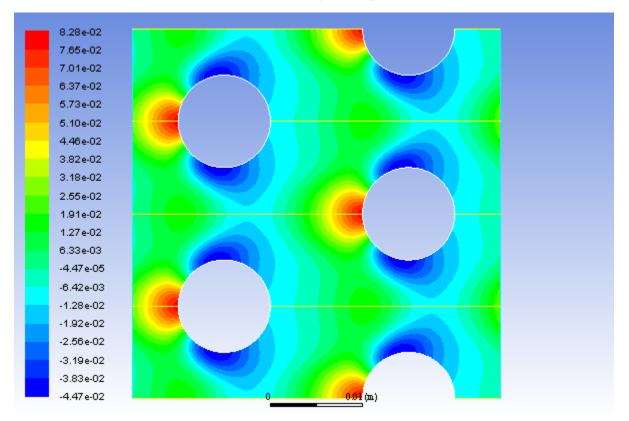
in the **Mirror Planes** selection list by clicking 🔽 in the upper right corner.

#### Note

There are four symmetry zones in the **Mirror Planes** selection list because the top and bottom symmetry planes in the domain are each composed of two symmetry zones, one on each side of the tube centered on the plane. It is also possible to generate the same display shown in Figure 4.4: Contours of Static Pressure with Symmetry (p. 222) by selecting just one of the symmetry zones on the top symmetry plane, and one on the bottom.

- ii. Click Apply and close the Views dialog box.
- iii. Translate the display of symmetry contours so that it is centered in the graphics window by using the left mouse button (Figure 4.4: Contours of Static Pressure with Symmetry (p. 222)).

Figure 4.4: Contours of Static Pressure with Symmetry



The pressure contours displayed in Figure 4.4: Contours of Static Pressure with Symmetry (p. 222) do not include the linear pressure gradient computed by the solver. Thus, the contours are periodic at the inlet and outflow boundaries.

2. Display filled contours of static temperature (Figure 4.5: Contours of Static Temperature (p. 223)).

Postprocessing  $\rightarrow$  Graphics  $\rightarrow$  Contours  $\rightarrow$  Edit...

Contours		×
Options	Contours of	
Filled	Temperature	•
Node Values	Static Temperature	-
Global Range Auto Range	Min (k)	Max (k)
Clip to Range	277.1682	400
Draw Profiles	Surfaces Filter Text	x
Coloring Banded Smooth Levels Setup 20 1	interior-15 periodic-9 symmetry-11 symmetry-13 symmetry-18 symmetry-24 wall-bottom New Surface Display Compute	E Close Help

- a. Select Temperature... and Static Temperature from the Contours of drop-down lists.
- b. Click **Display** and close the **Contours** dialog box.

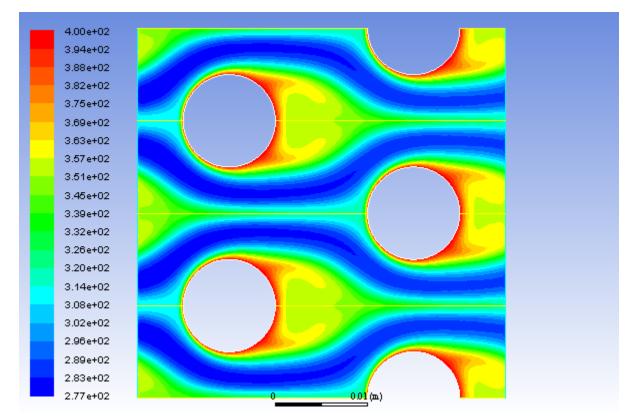


Figure 4.5: Contours of Static Temperature

The contours in Figure 4.5: Contours of Static Temperature (p. 223) reveal the temperature increase in the fluid due to heat transfer from the tubes. The hotter fluid is confined to the near-wall and wake regions, while a narrow stream of cooler fluid is convected through the tube bank.

3. Display the velocity vectors (Figure 4.6: Velocity Vectors (p. 225)).

#### Postprocessing $\rightarrow$ Graphics $\rightarrow$ Vectors $\rightarrow$ Edit...

Vectors		×
Options Global Range Global Range Glip to Range Glip to Range Glip to Scale Draw Mesh	Vectors of Velocity Color by Velocity Velocity Magnitude Min Max	•
Style arrow Scale Skip	Surfaces Filter Text	<b>-</b> x
2 0	interior-15 periodic-9 symmetry-11 symmetry-13 symmetry-18 symmetry-24	•
	Display Compute Close Help	

a. Enter 2 for Scale.

This will increase the size of the displayed vectors, making it easier to view the flow patterns.

- b. Retain the default selection of Velocity from the Vectors of drop-down list.
- c. Retain the default selection of Velocity... and Velocity Magnitude from the Color by drop-down lists.
- d. Click **Display** and close the **Vectors** dialog box.
- e. Zoom in on the upper right portion of one of the left tubes to get the display shown in (Figure 4.6: Velocity Vectors (p. 225)), by using the middle mouse button in the graphics window.

The magnified view of the velocity vector plot in Figure 4.6: Velocity Vectors (p. 225) clearly shows the recirculating flow behind the tube and the boundary layer development along the tube surface.

#### Figure 4.6: Velocity Vectors

1	114	
	1.31e-02	A A A A A A A A A A A A A A A A A A A
2	1.25e-02	
÷-	1.18e-02	
2	1.12e-02	
2	1.05e-02	
2	9.849-03	
2	9.196.03	
2	553×03	
2 - C	7.88403	
13	6 A Mall	
1	7.228-08	
2	-6.56e-03	
8/	5,81e-03	
8	5.25e-03	- 「「「「「」」、「「」」、「「」、「」、「」、「」、「」、「」、「」、「」、「」
6	4.60e-03	
6	3/94e-03	
2	3.28e-03	
2	2.63e-03	
2/	1.97e-03	
1	1.31e-03	and the second
16	6.58e-04	
$m_{\rm e}$	1.73e-06	0 0.002 (m)
<b>366</b>	1.100.00	

4. Create an isosurface on the periodic tube bank at x = 0.01 m (through the first column of tubes).

This isosurface and the ones created in the steps that follow will be used for the plotting of temperature profiles.

Iso-Surface	
Surface of Constant Mesh	From Surface Filter Text
X-Coordinate       Min (m)       0       0.04       Iso-Values (m)       0.01       New Surface Name       x-coordinate-9	interior-15 periodic-9 symmetry-11 symmetry-13 symmetry-24 From Zones Fiter Text fluid-16
Create Compute	Manage Close Help

**Setting Up Domain**  $\rightarrow$  Surface  $\rightarrow$  Create  $\rightarrow$  Iso-Surface...

- a. Select Mesh... and X-Coordinate from the Surface of Constant drop-down lists.
- b. Enter 0.01 for Iso-Values.
- c. Enter x=0.01m for New Surface Name.
- d. Click **Create**.
- 5. In a similar manner, create an isosurface on the periodic tube bank at x = 0.02 m (halfway between the two columns of tubes) named x=0.02m.
- 6. In a similar manner, create an isosurface on the periodic tube bank at x = 0.03 m (through the middle of the second column of tubes) named x=0.03m, and close the **Iso-Surface** dialog box.
- 7. Create an XY plot of static temperature on the three isosurfaces (Figure 4.7: Static Temperature at x=0.01, 0.02, and 0.03 m (p. 228)).

10001 10	Postprocessing	$\rightarrow$	$\textbf{Plots} \rightarrow$	XY	$\textbf{Plot} \rightarrow$	Edit
----------	----------------	---------------	------------------------------	----	-----------------------------	------

Solution XY Plot			
Options <ul> <li>Node Values</li> <li>Position on X Axis</li> <li>Position on Y Axis</li> <li>Write to File</li> <li>Order Points</li> </ul> File Data		Plot Direction X 0 Y 1 Z 0 Load File Free Data	Y Axis Function Temperature Static Temperature X Axis Function Direction Vector Surfaces Filter Text Symmetry-18 Symmetry-24 wall-bottom wall-top x=0.01m x=0.02m x=0.03m v
Plot Axes Curves Close Help			

a. Enter 0 for **X** and 1 for **Y** in the **Plot Direction** group box.

With a **Plot Direction** vector of (0, 1), ANSYS Fluent will plot the selected variable as a function of y. Since you are plotting the temperature profile on cross sections of constant x, the temperature varies with the y direction.

- b. Select Temperature... and Static Temperature from the Y-Axis Function drop-down lists.
- c. Select x=0.01m, x=0.02m, and x=0.03m in the Surfaces selection list.

Scroll down to find the **x=0.01m**, **x=0.02m**, and **x=0.03m** surfaces.

d. Click the **Curves...** button to open the **Curves - Solution XY Plot** dialog box.

This dialog box is used to define plot styles for the different plot curves.

Curves - So	lution XY Plot	<b>X</b>	
Curve # 0 🖨 Sample	Line Style Pattern Color foreground Weight 1	Marker Style Symbol + Color foreground Size 0.3	
Apply Close Help			

i. Select + from the **Symbol** drop-down list.

Scroll up to find the + item.

- ii. Click **Apply** to assign the + symbol to the x = 0.01 m curve.
- iii. Set the **Curve #** to 1 to define the style for the x = 0.02 m curve.
- iv. Select **x** from the **Symbol** drop-down list.

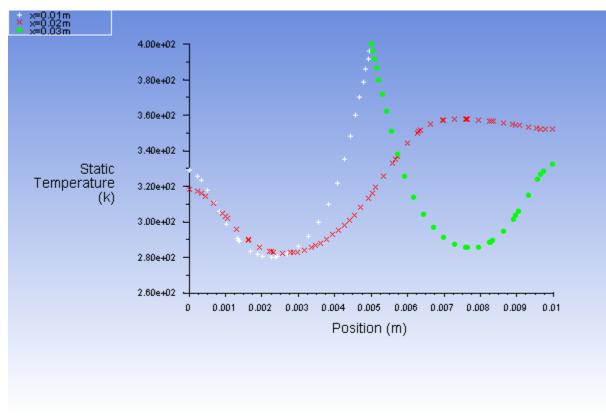
Scroll up to find the **x** item.

- v. Enter 0.5 for Size.
- vi. Click Apply and close the Curves Solution XY Plot dialog box.

Since you did not change the curve style for the x = 0.03 m curve, the default symbol will be used.

e. Click **Plot** and close the **Solution XY Plot** dialog box.





## 4.5. Summary

In this tutorial, periodic flow and heat transfer in a staggered tube bank were modeled in ANSYS Fluent. The model was set up assuming a known mass flow through the tube bank and constant wall temperatures. Due to the periodic nature of the flow and symmetry of the geometry, only a small piece of the full geometry was modeled. In addition, the tube bank configuration lent itself to the use of a hybrid mesh with quadrilateral cells around the tubes and triangles elsewhere.

The **Periodic Conditions** dialog box makes it easy to run this type of model with a variety of operating conditions. For example, different flow rates (and hence different Reynolds numbers) can be studied, or a different inlet bulk temperature can be imposed. The resulting solution can then be examined to extract the pressure drop per tube row and overall Nusselt number for a range of Reynolds numbers.

For additional details about modeling periodic heat transfer, see the Fluent User's Guide.

## 4.6. Further Improvements

This tutorial guides you through the steps to reach an initial solution. You may be able to obtain a more accurate solution by adapting the mesh. Mesh adaption can also ensure that the solution is independent of the mesh. These steps are demonstrated in Introduction to Using ANSYS Fluent: Fluid Flow and Heat Transfer in a Mixing Elbow (p. 121).

# **Chapter 5: Modeling External Compressible Flow**

This tutorial is divided into the following sections:

- 5.1. Introduction
- 5.2. Prerequisites
- 5.3. Problem Description
- 5.4. Setup and Solution
- 5.5. Summary
- 5.6. Further Improvements

## 5.1. Introduction

The purpose of this tutorial is to compute the turbulent flow past a transonic airfoil at a nonzero angle of attack. You will use the Spalart-Allmaras turbulence model.

This tutorial demonstrates how to do the following:

- Model compressible flow (using the ideal gas law for density).
- Set boundary conditions for external aerodynamics.
- Use the Spalart-Allmaras turbulence model.
- Use Full Multigrid (FMG) initialization to obtain better initial field values.
- Calculate a solution using the pressure-based coupled solver with the pseudo transient option.
- Use force and surface reports to check solution convergence.
- Check the near-wall mesh resolution by plotting the distribution of  $y^+$ .

# 5.2. Prerequisites

This tutorial is written with the assumption that you have completed one or more of the introductory tutorials found in this manual:

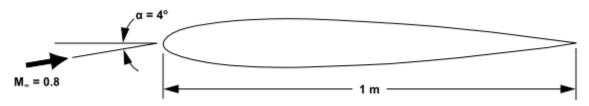
- Introduction to Using ANSYS Fluent in ANSYS Workbench: Fluid Flow and Heat Transfer in a Mixing Elbow (p. 1)
- Parametric Analysis in ANSYS Workbench Using ANSYS Fluent (p. 73)
- Introduction to Using ANSYS Fluent: Fluid Flow and Heat Transfer in a Mixing Elbow (p. 121)

and that you are familiar with the ANSYS Fluent tree and ribbon structure. Some steps in the setup and solution procedure will not be shown explicitly.

# **5.3. Problem Description**

The problem considers the flow around an airfoil at an angle of attack  $\alpha = 4^{\circ}$  and a free stream Mach number of 0.8 ( $M_{\infty}=0.8$ ). The flow is transonic, and has a fairly strong shock near the mid-chord (x / c=0.45) on the upper (suction) side. The chord length is 1 m. The geometry of the airfoil is shown in Figure 5.1: Problem Specification (p. 230).

#### Figure 5.1: Problem Specification



# 5.4. Setup and Solution

The following sections describe the setup and solution steps for this tutorial:

5.4.1. Preparation 5.4.2. Mesh 5.4.3. Solver 5.4.4. Models 5.4.5. Materials 5.4.6. Boundary Conditions 5.4.7. Operating Conditions 5.4.8. Solution 5.4.9. Postprocessing

# 5.4.1. Preparation

To prepare for running this tutorial:

- 1. Set up a working folder on the computer you will be using.
- 2. Go to the ANSYS Customer Portal, https://support.ansys.com/training.

#### Note

If you do not have a login, you can request one by clicking **Customer Registration** on the log in page.

- 3. Enter the name of this tutorial into the search bar.
- 4. Narrow the results by using the filter on the left side of the page.
  - a. Click ANSYS Fluent under Product.
  - b. Click 18.0 under Version.
- 5. Select this tutorial from the list.

- 6. Click the **external\_compressible\_R180.zip** link to download the input files.
- 7. Unzip external\_compressible\_R180.zip to your working folder.

The file airfoil.msh can be found in the external\_compressible folder created after unzipping the file.

8. Use Fluent Launcher to start the **2D** version of ANSYS Fluent.

Fluent Launcher displays your **Display Options** preferences from the previous session.

For more information about Fluent Launcher, see starting ANSYS Fluent using the Fluent Launcher in the Fluent Getting Started Guide.

- 9. Ensure that the **Display Mesh After Reading** and **Workbench Color Scheme** options are enabled.
- 10. Enable **Double Precision**.
- 11. Ensure Serial is selected under Processing Options.

### 5.4.2. Mesh

1. Read the mesh file airfoil.msh.

### **File** $\rightarrow$ Read $\rightarrow$ Mesh...

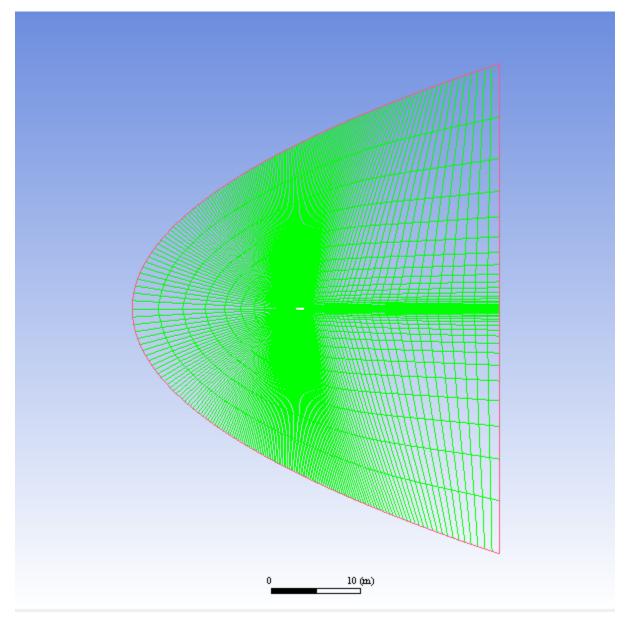
2. Check the mesh.



ANSYS Fluent will perform various checks on the mesh and will report the progress in the console. Make sure that the reported minimum volume is a positive number.

3. Examine the mesh (Figure 5.2: The Entire Mesh (p. 232) and Figure 5.3: Magnified View of the Mesh Around the Airfoil (p. 233)).

#### Figure 5.2: The Entire Mesh



Quadrilateral cells were used for this simple geometry because they can be stretched easily to account for different flow gradients in different directions. In the present case, the gradients normal to the airfoil wall are much greater than those tangent to the airfoil. Consequently, the cells near the surface have high aspect ratios. For geometries that are more difficult to mesh, it may be easier to create a hybrid mesh composed of quadrilateral and triangular cells.

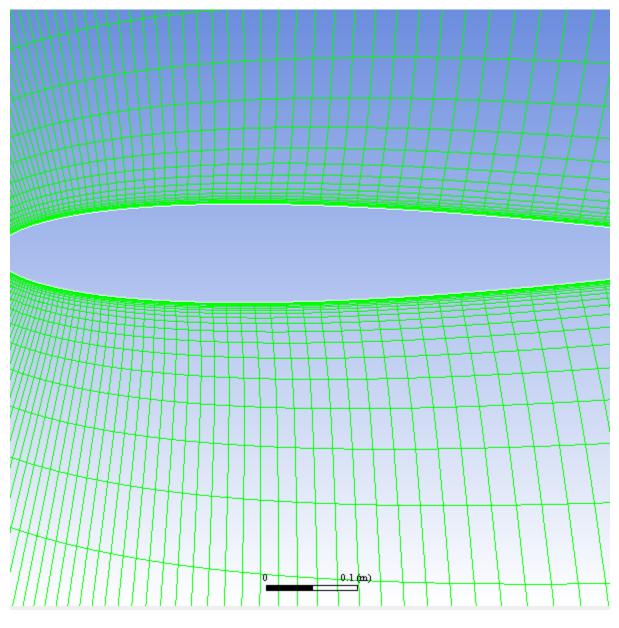


Figure 5.3: Magnified View of the Mesh Around the Airfoil

A parabola was chosen to represent the far-field boundary because it has no discontinuities in slope, enabling the construction of a smooth mesh in the interior of the domain.

#### Extra

You can use the right mouse button to probe for mesh information in the graphics window. If you click the right mouse button on any node in the mesh, information will be displayed in the ANSYS Fluent console about the associated zone, including the name of the zone. This feature is especially useful when you have several zones of the same type and you want to distinguish between them quickly.

## 5.4.3. Solver

1. Set the solver settings.

Setting Up Physics → Solver					
	Solver				
Time Type	Velocity Formulation	Operating Conditions			
<ul> <li>Steady</li> <li>Pressu</li> <li>Transient</li> <li>Densit</li> </ul>	ě	Reference Values			
Transient Density-Based	у-вазец 🔘 Кејаціче	Planar			

a. Retain the default selection of **Pressure-Based** from the **Type** list.

The pressure-based solver with the **Coupled** option for the pressure-velocity coupling is a good alternative to density-based solvers of ANSYS Fluent when dealing with applications involving high-speed aerodynamics with shocks. Selection of the coupled algorithm is made in the **Solution Methods** task page in the Solution step.

### 5.4.4. Models

1. Select the Spalart-Allmaras turbulence model.

100001 H0001 H	Setting l	Jp Physics $\rightarrow$	Models →	Viscous
----------------	-----------	--------------------------	----------	---------

I Viscous Model	<b>—</b>		
Model Inviscid Laminar Spalart-Allmaras (1 eqn) k-epsilon (2 eqn)	Model Constants Cb1 0.1355 Cb2 0.622		
<ul> <li>k-omega (2 eqn)</li> <li>Transition k-kl-omega (3 eqn)</li> <li>Transition SST (4 eqn)</li> <li>Reynolds Stress (5 eqn)</li> <li>Scale-Adaptive Simulation (SAS)</li> <li>Detached Eddy Simulation (DES)</li> </ul>	Cv1 7.1 Cw2 0.3		
<ul> <li>Spalart-Allmaras Production</li> <li>Vorticity-Based</li> <li>Strain/Vorticity-Based</li> </ul>	User-Defined Functions Turbulent Viscosity none		
Options Curvature Correction OK Cancel Help			

- a. Select Spalart-Allmaras (1eqn) in the Model list.
- b. Select Strain/Vorticity-Based in the Spalart-Allmaras Production list.
- c. Retain the default settings in the Model Constants group box.

d. Click **OK** to close the **Viscous Model** dialog box.

#### Note

The Spalart-Allmaras model is a relatively simple one-equation model that solves a modeled transport equation for the kinematic eddy (turbulent) viscosity. This embodies a relatively new class of one-equation models in which it is not necessary to calculate a length scale related to the local shear layer thickness. The Spalart-Allmaras model was designed specifically for aerospace applications involving wall-bounded flows and has been shown to give good results for boundary layers subjected to adverse pressure gradients.

### 5.4.5. Materials

The default **Fluid Material** is air, which is the working fluid in this problem. The default settings need to be modified to account for compressibility and variations of the thermophysical properties with temperature.

1. Set the properties for **air**, the default fluid material.

$\blacksquare Setup \rightarrow Materials \rightarrow Fluids \rightarrow air \stackrel{\square}{\hookrightarrow} Ec$	dit
--	-----

Create/Edit Materials					×
Name		Material Type			Order Materials by
air		fluid		•	Name
Chemical Formula		FLUENT Fluid Mat	erials		Chemical Formula
		air		•	FLUENT Database
		Mixture			User-Defined Database
		none		$\overline{\mathbf{v}}$	
Properties					
Density (kg/m3)	ideal-gas		• Edit	Î	
Cp (Specific Heat) (j/kg-k)	constant		▼ Edit		
Thermal Conductivity (w/m-k)	constant		▼ Edit		
	0.0242				
Viscosity (kg/m-s)	sutherland		▼ Edit		
				-	
(	Change/Create	Delete	Close	Help	

a. Select ideal-gas from the Density drop-down list.

The **Energy Equation** will be enabled.

b. Select sutherland from the Viscosity drop-down list to open the Sutherland Law dialog box.

💶 Sutherland Law 📃	3
Methods	
<ul> <li>Two Coefficient Method (SI Units Only)</li> <li>Three Coefficient Method</li> </ul>	
Reference Viscosity, mu0 (kg/m-s) 1.716e-05	^
Reference Temperature, T0 (k) 273.11	
Effective Temperature, S (k) 110.56	÷
OK Cancel Help	

Scroll down the **Viscosity** drop-down list to find **sutherland**.

- i. Retain the default selection of Three Coefficient Method in the Methods list.
- ii. Click OK to close the Sutherland Law dialog box.

The Sutherland law for viscosity is well suited for high-speed compressible flows.

- c. Click Change/Create to save these settings.
- d. Close the Create/Edit Materials dialog box.

While **Density** and **Viscosity** have been made temperature-dependent, **Cp** (**Specific Heat**) and **Thermal Conductivity** have been left constant. For high-speed compressible flows, thermal dependency of the physical properties is generally recommended. For simplicity, **Thermal Conductivity** and **Cp** (**Specific Heat**) are assumed to be constant in this tutorial.

## 5.4.6. Boundary Conditions

Setup → ♀Boundary Conditions

Boun	dary Condit	ions	
Zone	Filter Text		
	or-1 ure-far-field-1 bottom		
wall-1	top		
Phase mixtu		isure-far-field  ID 11	
	Edit	Copy Profiles	
	ameters lay Mesh	Operating Conditions Periodic Conditions	
Help	0		

1. Set the boundary conditions for **pressure-far-field-1**.

E Setup →	Boundary Conditions	→ E pressure-far-f	ield-1 → Edit
-----------	---------------------	--------------------	---------------

Pressure Far-Field					×
Zone Name pressure-far-field-1					
Momentum Thermal	Radiation	Species	Potential	UDS	DPM
Gauge Pressure (pascal)	0		constant		•
Mach Number	0.8		constant		•
X-Component of Flow Direction	0.997564		constant		•
Y-Component of Flow Direction	0.069756		constant		•
Turbulence					
Specification Method Tur	bulent Viscos	ity Ratio			<b>•</b>
Turbulent Viscosity Ratio	10		constant		<b>-</b>
L	OK Can	cel Help			

a. Retain the default value of 0 Pa for Gauge Pressure.

#### Note

The gauge pressure in ANSYS Fluent is always relative to the operating pressure, which is defined in a separate input (see below).

- b. Enter 0.8 for Mach Number.
- c. Enter 0.997564 and 0.069756 for the X-Component of Flow Direction and Y-Component of Flow Direction, respectively.

These values are determined by the 4° angle of attack:  $\cos 4^{\circ} \approx 0.997564$  and  $\sin 4^{\circ} \approx 0.069756$ .

- d. Retain **Turbulent Viscosity Ratio** from the **Specification Method** drop-down list in the **Turbulence** group box.
- e. Retain the default value of 10 for **Turbulent Viscosity Ratio**.

The viscosity ratio should be between 1 and 10 for external flows.

f. Click the **Thermal** tab and retain the default value of 300 K for **Temperature**.

Pressure Far-Field					×
Zone Name					
pressure-far-field-1					
Momentum Thermal	Radiation	Species	Potential	UDS	DPM
Temperature (k) 300		constant		•	
	OK Car	ncel Help			

i. A. Click **OK** to close the **Pressure Far-Field** dialog box.

### 5.4.7. Operating Conditions

1. Set the operating pressure.

E Setup  $\rightarrow \textcircled{P}$  Boundary Conditions  $\rightarrow$  Operating Conditions...

Operating Conditions	×
Pressure	Gravity
Operating Pressure (pascal)	Gravity
101325 P	
Reference Pressure Location	
X (m) 0	
Y (m) 0	
Z (m) 0	
OK Cancel Help	

The **Operating Conditions** dialog box can also be accessed from the **Cell Zone Conditions** task page.

a. Retain the default value of 101325 Pa for **Operating Pressure**.

The operating pressure should be set to a meaningful mean value in order to avoid round-off errors. The absolute pressure must be greater than zero for compressible flows. If you want to specify boundary conditions in terms of absolute pressure, you can make the operating pressure zero.

b. Click **OK** to close the **Operating Conditions** dialog box.

For information about setting the operating pressure, see the Fluent User's Guide.

### 5.4.8. Solution

1. Set the solution parameters.



Solution Methods	
Pressure-Velocity Coupling	
Scheme	
Coupled	•
Spatial Discretization	
Gradient	*
Least Squares Cel Based 🔹	
Pressure	
Second Order 👻	
Density	Ξ
Second Order Upwind	
Momentum	
Second Order Upwind	
Modified Turbulent Viscosity	
Second Order Upwind	-
Transient Formulation	_
Non-Iterative Time Advancement	
Frozen Flux Formulation	
V Pseudo Transient	
Warped-Face Gradient Correction	
High Order Term Relaxation Options	
Default	
Нер	

- a. Select Coupled from the Scheme drop-down list in the Pressure-Velocity Coupling group box.
- b. Retain the default selection of **Least Squares Cell Based** from the **Gradient** drop-down list in the **Spatial Discretization** group box.
- c. Retain the default selection of Second Order from the Pressure drop-down list.
- d. Select Second Order Upwind from the Modified Turbulent Viscosity drop-down list.
- e. Enable Pseudo Transient.

The Pseudo Transient option enables the pseudo transient algorithm in the coupled pressure-based solver. This algorithm effectively adds an unsteady term to the solution equations in order to improve stability and convergence behavior. Use of this option is recommended for general fluid flow problems.

2. Set the solution controls.



Solution Controls	
Pseudo Transient Explicit Relaxation Factors	
Pressure	
0.5	
Momentum	
0.5	=
Density	-
0.5	
Body Forces	
1	
Modified Turbulent Viscosity	
0.9	
Turkelank Manaaka	
Default	
Equations Limits Advanced	
Help	

a. Enter 0.5 for **Density** in the **Pseudo Transient Explicit Relaxation Factors** group box.

Under-relaxing the density factor is recommended for high-speed compressible flows.

b. Enter 0.9 for Modified Turbulent Viscosity.

Larger under-relaxation factors (that is, closer to 1) will generally result in faster convergence. However, instability can arise that may need to be eliminated by decreasing the under-relaxation factors.

3. Enable residual plotting during the calculation.

**Solving**  $\rightarrow$  Reports  $\rightarrow$  Residuals...

Residual Monitors					<b>-X</b>
Options          Image: Print to Console         Image: Plot	Equations Residual continuity	Monitor	Check Convergence	Absolute Criteria	
Window       1     Curves       Xess	x-velocity y-velocity		<b>V</b>	0.001	II
Iterations to Plot	energy Residual Values			1e-06 Convergence Cr	-
Iterations to Store	Normalize		Iterations 5	absolute	▼
	Scale	al Scale			
OK Plot	Renormaliz	e (	Cancel Hel	p	

- a. Ensure that **Plot** is enabled in the **Options** group box and click **OK** to close the **Residual Monitors** dialog box.
- 4. Initialize the solution.

Solution $\rightarrow $ Solution Initialization
Solution Initialization Initialization Methods <ul> <li>Hybrid Initialization</li> <li>Standard Initialization</li> </ul>
More Settings
Patch

More Settings Initialize
Patch
Reset DPM Sources Reset Statistics
Help

- a. Retain the default selection of Hybrid Initialization from the Initialization Methods group box.
- b. Click **Initialize** to initialize the solution.

c. Run the Full Multigrid (FMG) initialization.

FMG initialization often facilitates an easier start-up, where no CFL (Courant Friedrichs Lewy) ramping is necessary, thereby reducing the number of iterations for convergence.

- i. Press **Enter** in the console to get the command prompt (>).
- ii. Enter the text commands and input responses as shown in the boxes. Accept the default values by pressing **Enter** when no input response is given:

```
solve/initialize/set-fmg-initialization
Customize your FMG initialization:
  set the number of multigrid levels [5]
  set FMG parameters on levels ..
   residual reduction on level 1 is: [0.001]
   number of cycles on level 1 is: [10] 100
   residual reduction on level 2 is: [0.001]
   number of cycles on level 2 is: [50] 100
   residual reduction on level 3 is: [0.001]
   number of cycles on level 3 is: [100]
   residual reduction on level 4 is: [0.001]
   number of cycles on level 4 is: [500]
   residual reduction on level 5 [coarsest grid] is: [0.001]
   number of cycles on level 5 is: [500]
 * FMG customization complete
   set FMG courant-number [0.75]
  enable FMG verbose? [no] yes
  solve/initialize/fmg-initialization
Enable FMG initialization? [no] yes
```

#### Note

Whenever FMG initialization is performed, it is important to inspect the FMG initialized flow field using the postprocessing tools of ANSYS Fluent. Monitoring the normalized residuals, which are plotted in the console window, will give you an idea of the convergence of the FMG solver. You should notice that the value of the normalized residuals decreases. For information about FMG initialization, including convergence strategies, see the Fluent User's Guide.

5. Save the case and data files (airfoil.cas.gz and airfoil.dat.gz).

```
File → Write → Case & Data...
```

It is good practice to save the case and data files during several stages of your case setup.

6. Start the calculation by requesting 50 iterations.

### Solving → Run Calculation

a. Enter 50 for No. of Iterations.

#### b. Click Calculate.

By performing some iterations before setting up the force reports, you will avoid large initial transients in the report plots. This will reduce the axes range and make it easier to judge the convergence.

7. Set the reference values that are used to compute the lift, drag, and moment coefficients.

# E Setup → ♥ Reference Values

The reference values are used to non-dimensionalize the forces and moments acting on the airfoil. The dimensionless forces and moments are the lift, drag, and moment coefficients.

Reference Values	
Compute from	
pressure-far-field-1	•
Reference Values	
Area (m2)	1
Density (kg/m3)	1.176674
Depth (m)	1
Enthalpy (j/kg)	40412.25
Length (m)	1
Pressure (pascal)	0
Temperature (k)	300
Velocity (m/s)	277.6702
Viscosity (kg/m-s)	1.7894e-05
Ratio of Specific Heats	1.4
Reference Zone	
fluid-16	•
Help	

a. Select **pressure-far-field-1** from the **Compute from** drop-down list.

ANSYS Fluent will update the **Reference Values** based on the boundary conditions at the far-field boundary.

8. Create a force report definition to plot and write the drag coefficient for the walls of the airfoil.

100001 10000 10	Solving $\rightarrow$	Reports →	Definitions	→ New →	Force Re	port →	Drag
-----------------	-----------------------	-----------	-------------	---------	----------	--------	------

Name		×
cd-1		
Options	Report Output Type     Drag Coefficient     Drag Force	
Per Zone Average Over(Iterations)	Wall Zones Filter Text	
1	wall-top	
Force Vector		
X Y Z 0.9976 0.06976 1		
0.9976 0.06976 1		
Report Files [0/0]		
Report Plots [0/0]		
Create		
Report File		
Report File     Report Plot		
Report File     Report Plot     Frequency 1		
Report File     Report Plot     Frequency 1     Print to Console		
Report File     Report Plot     Frequency 1		

- a. Enter cd-1 for Name.
- b. Make sure that Drag Coefficient is selected in the Report Output Type group box.
- c. In the Create group box, enable Report Plot.
- d. Enable **Report File** to save the report history to a file.

#### Note

If you do not enable the **Report File** option, the history information will be lost when you exit ANSYS Fluent.

- e. Select wall-bottom and wall-top in the Wall Zones selection list.
- f. Enter 0.9976 for **X** and 0.06976 for **Y** in the **Force Vector** group box.

These **X** and **Y** values ensure that the drag coefficient is calculated parallel to the free-stream flow, which is  $4^{\circ}$  off of the global coordinates.

- g. Click **OK** to close the **Drag Report Definition** dialog box.
- 9. Similarly, create a force report definition for the lift coefficient.

**Solving**  $\rightarrow$  Reports  $\rightarrow$  Definitions  $\rightarrow$  New  $\rightarrow$  Force Report  $\rightarrow$  Lift...

Lift Report Definition				<b>×</b>
Name				
d-1				
Options		Report Output Type     Lift Coefficient     Lift Force		
Per Zone		Wall Zones Filter Text		x- 5- 5- 5-
Average Over(Iterations)		wall-bottom		
1		wall-bottom wall-top		
Force Vector				
Х Ү Z				
-0.0698 0.9976 1				
Report Files [0/1]		1		
cd-1-rfile Report Plots [0/1] cd-1-rplot				
Report File				
Report Plot				
Frequency 1				
Print to Console				
Create Output Parameter				
	ОК	Compute Cancel Hel	•	

Enter the values for **X** and **Y** shown in the Lift Report Definition dialog box.

The **X** and **Y** values shown ensure that the lift coefficient is calculated normal to the free-stream flow, which is  $4^{\circ}$  off of the global coordinates.

10. In a similar manner, create a force report definition for the moment coefficient.

Solving  $\rightarrow$  Reports  $\rightarrow$  Definitions  $\rightarrow$  New  $\rightarrow$  Force Report  $\rightarrow$  Moment...

lame cm-1			
Options			
opcono		Wall Zones Filter Text	
		wall-bottom	
		wall-top	
Per Zone			
Average Over(Iteratio			
1	۵		
Moment Center			
X (m) Y (m)	Z (m)		
0.25 0	1		
Moment Axis			
X Y	Z		
0 0	1		
Report Files [0/2] cd-1-rfile cl-1-rfile		x	
cd-1-rfile			
cd-1-rfile			
cd-1-rfile cl-1-rfile Report Plots [0/2]			
cd-1-rfile cl-1-rfile			
cd-1-rfile cl-1-rfile Report Plots [0/2] cd-1-rplot			
cd-1-rfile cl-1-rfile Report Plots [0/2] cd-1-rplot cl-1-rplot			
cd-1-rfile cl-1-rfile Report Plots [0/2] cd-1-rplot cl-1-rplot Create			
cd-1-rfile cl-1-rfile Report Plots [0/2] cd-1-rplot cl-1-rplot Create V Report File			
cd-1-rfile cl-1-rfile Report Plots [0/2] cd-1-rplot cl-1-rplot Create Report File Report Plot			
cd-1-rfile cl-1-rfile Report Plots [0/2] cd-1-rplot cl-1-rplot Create V Report File Report Plot Frequency 1			
cd-1-rfile cl-1-rfile Report Plots [0/2] cd-1-rplot cl-1-rplot cl-1-rplot V Report File V Report Plot Frequency 1 +			
cd-1-rfile cl-1-rfile Report Plots [0/2] cd-1-rplot cl-1-rplot Create V Report File Report Plot Frequency 1			

Enter the values for the **Moment Center** and **Moment Axis** shown in the **Moment Report Definition** dialog box.

11. Display filled contours of pressure overlaid with the mesh in preparation for creating a surface report definition (Figure 5.4: Pressure Contours After 50 Iterations (p. 249) and Figure 5.5: Magnified View of Pressure Contours Showing Wall-Adjacent Cells (p. 250)).

Postprocessing  $\rightarrow$  Graphics  $\rightarrow$  Contours  $\rightarrow$  Edit...

Contours	× (
Options Filed	Contours of Pressure
<ul> <li>Node Values</li> <li>Global Range</li> </ul>	Static Pressure 🗸
Auto Range Clip to Range	Min Max 0 0
Draw Profiles Traw Mesh	Surfaces Filter Text
Coloring Banded Smooth Levels Setup	interior-1 pressure-far-field-1 wall-bottom wall-top
20 🗣 1 🌩	New Surface 🔻
	Display Compute Close Help

- a. Enable **Filled** in the **Options** group box.
- b. Enable Draw Mesh to open the Mesh Display dialog box.

💶 Mesh Display	<b>×</b>
Options Edge Type Nodes All Edges Feature Faces Outline Overset	Surfaces Filter Text
Shrink FactorFeature Angle020	
Outline Interior	
Adjacency	New Surface 🔻
[	Display Colors Close Help

- i. Retain the default settings.
- ii. Close the **Mesh Display** dialog box.
- c. Click **Display** and close the **Contours** dialog box.

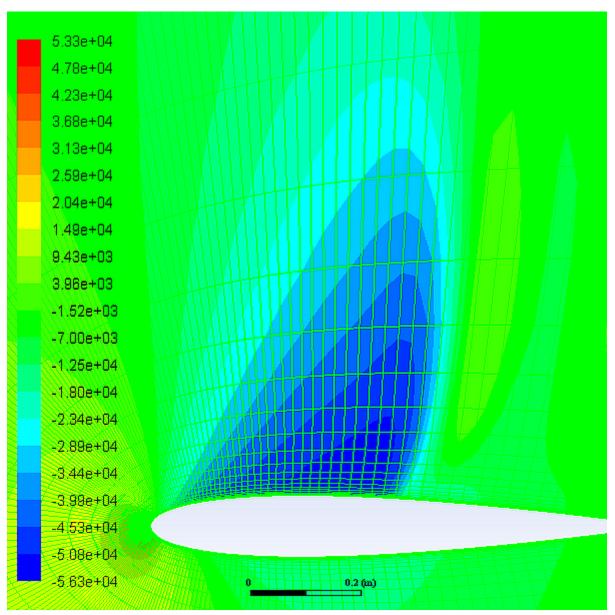


Figure 5.4: Pressure Contours After 50 Iterations

The shock is clearly visible on the upper surface of the airfoil, where the pressure jumps to a higher value downstream of the low pressure area.

### Note

The color indicating a high pressure area near the leading edge of the airfoil is obscured by the overlaid green mesh. To view this contour, simply disable the **Draw Mesh** option in the **Contours** dialog box and click **Display**.

d. Zoom in on the shock wave, until individual cells adjacent to the upper surface (**wall-top** boundary) are visible, as shown in Figure 5.5: Magnified View of Pressure Contours Showing Wall-Adjacent Cells (p. 250).

<b>5</b> .33e+04				
4.78e+04				
4.23e+04				
3.68e+04				
3.13e+04				
2.59e+04				
2.04e+04				
1.49e+04				
9.43e+03				
3.96e+03				
-1.52e+03				
-7.00e+03				
-1.25e+04				
-1.80e+04				
-2.34e+04				
-2.89e+04				
-3.44e+04				
-3.99e+04				
-4:53e+04				
-5.08e+04				
5.63e+04	0	0.0	5 (m)	

Figure 5.5: Magnified View of Pressure Contours Showing Wall-Adjacent Cells

The magnified region contains cells that are just downstream of the shock and adjacent to the upper surface of the airfoil. In the following step, you will create a point surface inside a wall-adjacent cell, which you will use to create a surface report definition.

12. Create a point surface just downstream of the shock wave.

Postprocessing  $\rightarrow$  Surface  $\rightarrow$  Create  $\rightarrow$  Point...

Point Surface	<b>×</b>
Options	Coordinates
Point Tool	x0 (m) 0.53
Reset	y0 (m) 0.051
	z0 (m) 0
Select New Surface Name	Point with Mouse
point-4	
Create	lanage Close Help

- a. Enter 0.53 m for **x0** and 0.051 m for **y0** in the **Coordinates** group box.
- b. Retain the default entry of point-4 for New Surface Name.
- c. Click **Create** and close the **Point Surface** dialog box.

#### Note

You have entered the exact coordinates of the point surface so that your convergence history will match the plots and description in this tutorial. In general, however, you will not know the exact coordinates in advance, so you will need to select the desired location in the graphics window. You do not have to apply the following instructions at this point in the tutorial; they are added here for your information:

- a. In the **Point Surface** dialog box, click the **Select Point with Mouse** button. A **Working** dialog box will open telling you to "Click on a location in the graphics window with the MOUSE-PROBE mouse button."
- b. Position the mouse pointer at a point located inside one of the cells adjacent to the upper surface (**wall-top** boundary), downstream of the shock (see Figure 5.6: Pressure Contours after Creating a Point with the Mouse (p. 252)).
- c. Click the right mouse button.
- d. Click Create to create the point surface and then close the Point Surface dialog box.

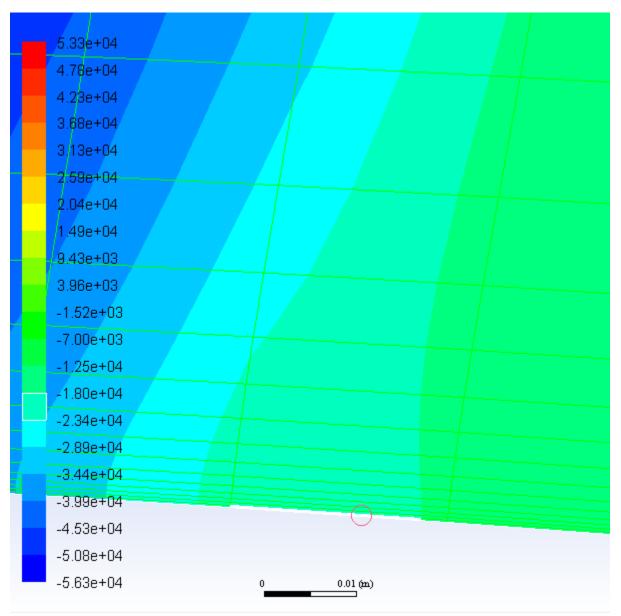


Figure 5.6: Pressure Contours after Creating a Point with the Mouse

13. Enable residual plotting during the calculation.

**Solving**  $\rightarrow$  Reports  $\rightarrow$  Residuals...

Residual Monitors			<b>×</b>
Options          Iterations to Plot	Equations Residual continuity x-velocity y-velocity energy	Monitor	
1000	Residual Values	Iterations	Convergence Criterion
1000 🖨	Scale Compute Loca Plot Renormali		Convergence Conditions

- a. Ensure that **Plot** is enabled in the **Options** group box.
- b. Select **none** from the **Convergence Criterion** drop-down list so that automatic convergence checking does not occur.
- c. Click **OK** to close the **Residual Monitors** dialog box.
- 14. Create a surface report definition for tracking the velocity magnitude value at the point created in the previous step.

Since the drag, lift, and moment coefficients are global variables, indicating certain overall conditions, they may converge while local conditions at specific points are still varying from one iteration to the next. To account for this, create a report definition at a point (just downstream of the shock) where there is likely to be significant variation, and monitor the value of the velocity magnitude.

Solving  $\rightarrow$  Reports  $\rightarrow$  Definitions  $\rightarrow$  New  $\rightarrow$  Surface Report  $\rightarrow$  Vertex Average...

Surface Report Definition	<b>—</b> ×-
Name	Report Type
surf-mon-1	Vertex Average 🔹
Options	Custom Vectors
	Vectors of
Per Surface	<b></b>
Average Over	Custom Vectors
1	
	Field Variable
Report Files [0/3]	Velocity 🔻
cd-1-rfile	Velocity Magnitude
cl-1-rfile	
cm-1-rfile	Surfaces Filter Text
	interior-1
Report Plots [0/0]	point-4
Report Plots [0/0]	
	wall-bottom
	wall-top
Create	
Report File	
Report Plot	
Frequency 1	
V Print to Console	
Create Output Parameter	New Surface 🔻
OK	npute Cancel Help
	.#

- a. Enter **surf-mon-1** for **Name**.
- b. Select Velocity... and Velocity Magnitude from the Field Variable drop-down list.
- c. Select **point-4** in the **Surfaces** selection list.
- d. In the Create group box, enable Report File, Report Plot and Print to Console.
- e. Click OK to close the Surface Report Definition dialog box.
- 15. Save the case and data files (airfoil-1.cas.gz and airfoil-1.dat.gz).

**File**  $\rightarrow$  Write  $\rightarrow$  Case & Data...

- 16. Continue the calculation for 200 more iterations.
  - Solving  $\rightarrow$  Run Calculation  $\rightarrow$  Calculate

250

The force reports (Figure 5.8: Drag Coefficient Convergence History (p. 256) and Figure 5.9: Lift Coefficient Convergence History (p. 257)) show that the case is converged after approximately 200 iterations.

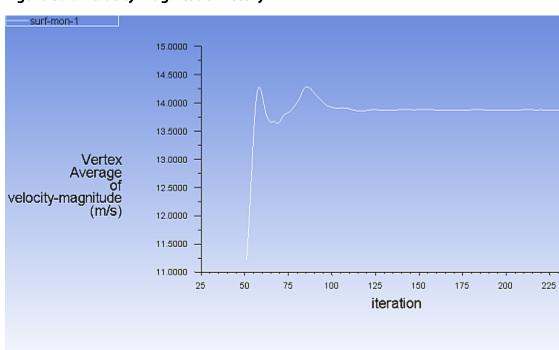
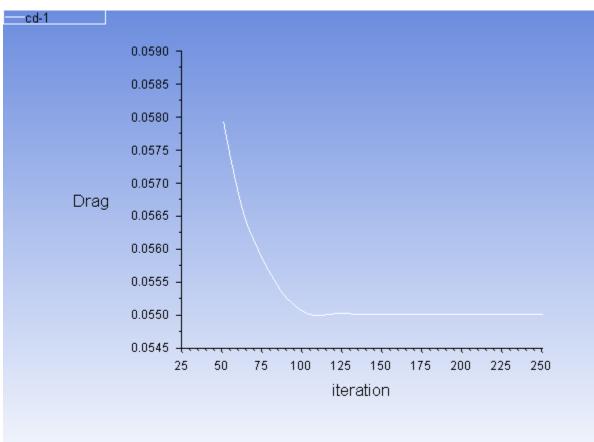


Figure 5.7: Velocity Magnitude History





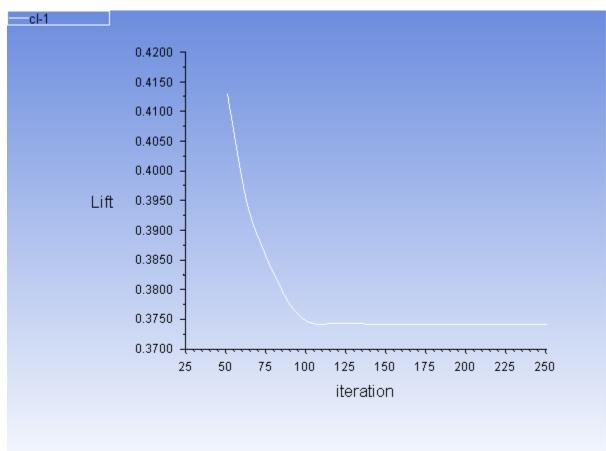
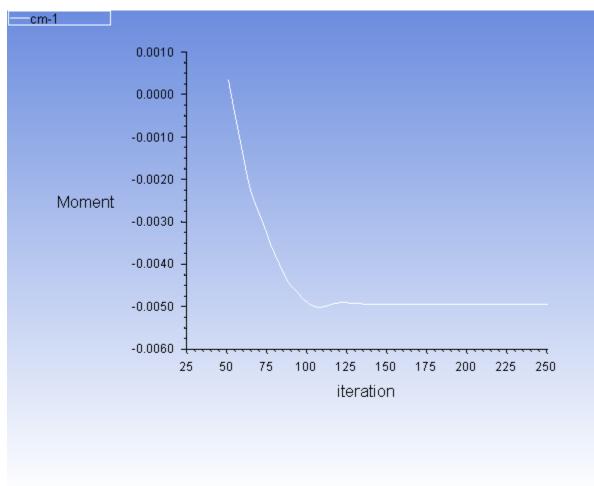


Figure 5.9: Lift Coefficient Convergence History





17. Save the case and data files (airfoil-2.cas.gz and airfoil-2.dat.gz).

**File**  $\rightarrow$  Write  $\rightarrow$  Case & Data...

## 5.4.9. Postprocessing

1. Plot the  $y^+$  distribution on the airfoil (Figure 5.11: XY Plot of y+ Distribution (p. 260)).

Postprocessing  $\rightarrow$  Plots  $\rightarrow$  XY Plot  $\rightarrow$  Edit...

Solution XY Plot					
Options Node Values Position on X Axis Position on Y Axis Vrite to File Order Points File Data		Plot Direction X 1 Y 0 Z 0	Y Axis Function Turbulence Wall Yplus X Axis Function Direction Vector Surfaces Filter Text interior-1 point-4 pressure-far-field-1 wall-bottom wall-top New Surface		
Plot Axes Curves Close Help					

- a. Disable Node Values in the Options group box.
- b. Select Turbulence... and Wall Yplus from the Y Axis Function drop-down list.

Wall Yplus is available only for cell values.

- c. Select wall-bottom and wall-top in the Surfaces selection list.
- d. Click Plot and close the Solution XY Plot dialog box.

#### Note

The values of  $y^+$  are dependent on the resolution of the mesh and the Reynolds number of the flow, and are defined only in wall-adjacent cells. The value of  $y^+$  in the wall-adjacent cells dictates how wall shear stress is calculated. When you use the Spalart-Allmaras model, you should check that  $y^+$  of the wall-adjacent cells is either very small (on the order of  $y^+=1$ ), or approximately 30 or greater. Otherwise, you should modify your mesh.

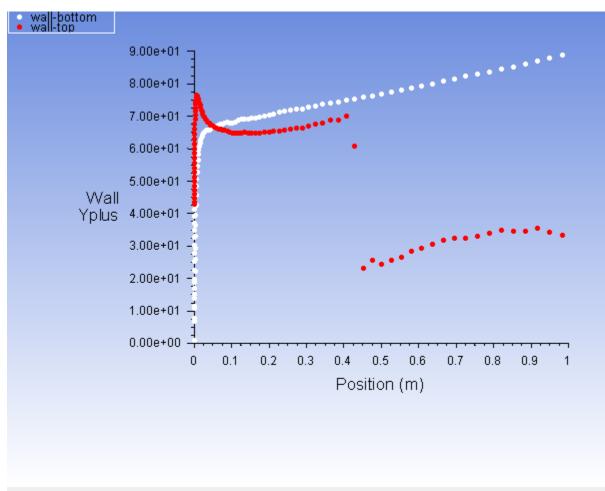
The equation for  $y^+$  is

$$y^{+} = \frac{y}{\mu} \sqrt{\rho \tau_{w}}$$
(5.1)

where y is the distance from the wall to the cell center,  $\mu$  is the molecular viscosity,  $\rho$  is the density of the air, and  $\tau_w$  is the wall shear stress.

Figure 5.11: XY Plot of y+ Distribution (p. 260) indicates that, except for a few small regions (notably at the shock and the trailing edge),  $y^+>30$  and for much of these regions it does not drop significantly below 30. Therefore, you can conclude that the near-wall mesh resolution is acceptable.





2. Display filled contours of Mach number (Figure 5.12: Contour Plot of Mach Number (p. 261)).

Postprocessing  $\rightarrow$  Graphics  $\rightarrow$  Contours  $\rightarrow$  Edit...

- a. Ensure Filled is enabled in the Options group box.
- b. Select Velocity... and Mach Number from the Contours of drop-down list.
- c. Click **Display** and close the **Contours** dialog box.
- d. Zoom in on the region around the airfoil, as shown in Figure 5.12: Contour Plot of Mach Number (p. 261).

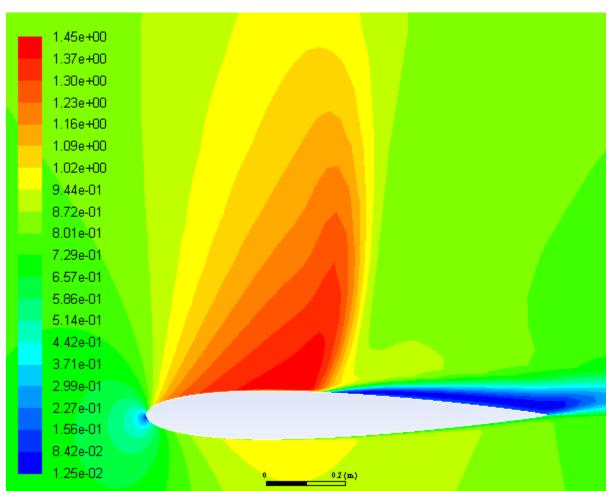


Figure 5.12: Contour Plot of Mach Number

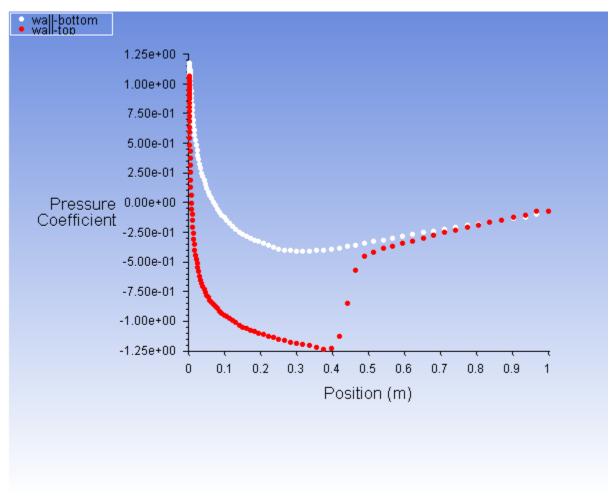
Note the discontinuity, in this case a shock, on the upper surface of the airfoil in Figure 5.12: Contour Plot of Mach Number (p. 261) at about  $x / c \approx 0.45$ .

3. Plot the pressure distribution on the airfoil (Figure 5.13: XY Plot of Pressure (p. 262)).

Postprocessing  $\rightarrow$  Plots  $\rightarrow$  XY Plot  $\rightarrow$  Edit...

- a. Enable Node Values.
- b. Select Pressure... and Pressure Coefficient from the Y Axis Function drop-down lists.
- c. Click Plot.

#### Figure 5.13: XY Plot of Pressure

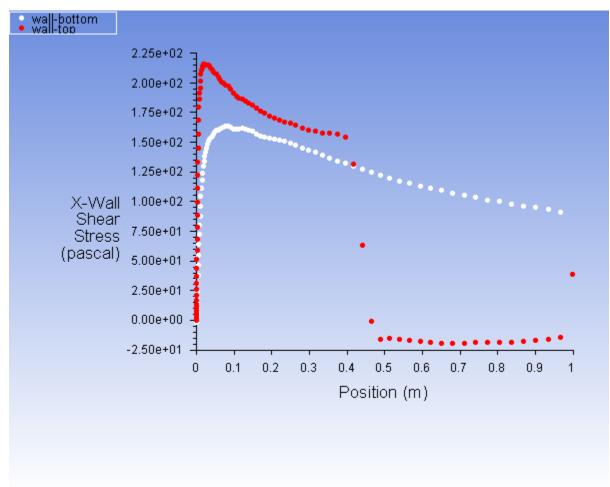


Notice the effect of the shock wave on the upper surface in Figure 5.13: XY Plot of Pressure (p. 262).

- 4. Plot the *x* component of wall shear stress on the airfoil surface (Figure 5.14: XY Plot of x Wall Shear Stress (p. 263)).
  - a. Disable Node Values.
  - b. Select Wall Fluxes... and X-Wall Shear Stress from the Y Axis Function drop-down lists.
  - c. Click Plot and close the Solution XY Plot dialog box.

As shown in Figure 5.14: XY Plot of x Wall Shear Stress (p. 263), the large, adverse pressure gradient induced by the shock causes the boundary layer to separate. The point of separation is where the wall shear stress vanishes. Flow reversal is indicated here by negative values of the x component of the wall shear stress.



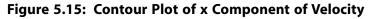


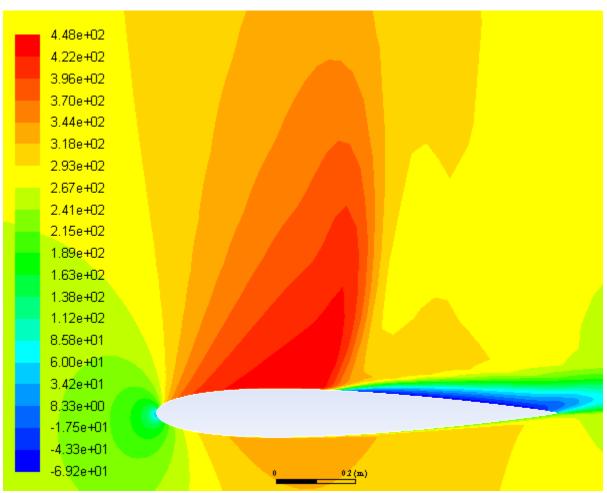
5. Display filled contours of the *x* component of velocity (Figure 5.15: Contour Plot of x Component of Velocity (p. 264)).

Postprocessing  $\rightarrow$  Graphics  $\rightarrow$  Contours  $\rightarrow$  Edit...

- a. Ensure Filled is enabled in the Options group box.
- b. Select Velocity... and X Velocity from the Contours of drop-down lists.
- c. Click **Display** and close the **Contours** dialog box.

Note the flow reversal downstream of the shock in Figure 5.15: Contour Plot of x Component of Velocity (p. 264).





6. Plot velocity vectors (Figure 5.16: Plot of Velocity Vectors Downstream of the Shock (p. 265)).

Postprocessing  $\rightarrow$  Graphics  $\rightarrow$  Vectors  $\rightarrow$  Edit...

- a. Enter 15 for Scale.
- b. Click **Display** and close the **Vectors** dialog box.
- c. Zoom in on the flow above the upper surface at a point downstream of the shock, as shown in Figure 5.16: Plot of Velocity Vectors Downstream of the Shock (p. 265).

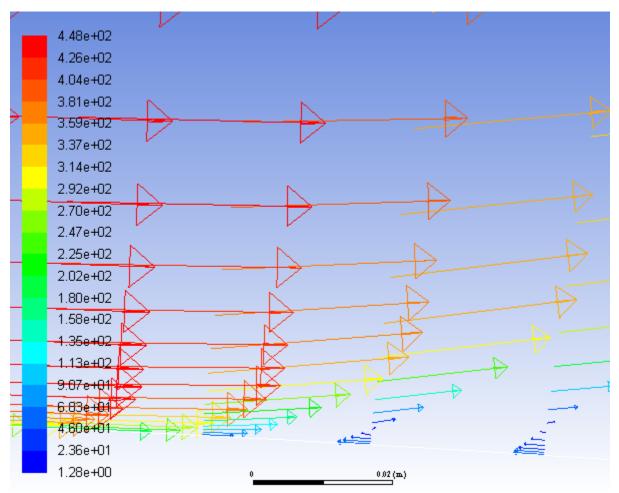


Figure 5.16: Plot of Velocity Vectors Downstream of the Shock

Flow reversal is clearly visible in Figure 5.16: Plot of Velocity Vectors Downstream of the Shock (p. 265).

# 5.5. Summary

This tutorial demonstrated how to set up and solve an external aerodynamics problem using the pressurebased coupled solver with pseudo transient under-relaxation and the Spalart-Allmaras turbulence model. It showed how to monitor convergence using force and surface report definitions, and demonstrated the use of several postprocessing tools to examine the flow phenomena associated with a shock wave.

# 5.6. Further Improvements

This tutorial guides you through the steps to reach an initial solution. You may be able to obtain a more accurate solution by using an appropriate higher-order discretization scheme and by adapting the mesh. Mesh adaption can also ensure that the solution is independent of the mesh. These steps are demonstrated in Introduction to Using ANSYS Fluent: Fluid Flow and Heat Transfer in a Mixing Elbow (p. 121).

# **Chapter 6: Modeling Transient Compressible Flow**

This tutorial is divided into the following sections:

- 6.1. Introduction
- 6.2. Prerequisites
- 6.3. Problem Description
- 6.4. Setup and Solution
- 6.5. Summary
- 6.6. Further Improvements

# 6.1. Introduction

In this tutorial, ANSYS Fluent's density-based implicit solver is used to predict the time-dependent flow through a two-dimensional nozzle. As an initial condition for the transient problem, a steady-state solution is generated to provide the initial values for the mass flow rate at the nozzle exit.

This tutorial demonstrates how to do the following:

- Calculate a steady-state solution (using the density-based implicit solver) as an initial condition for a transient flow prediction.
- Define a transient boundary condition using a user-defined function (UDF).
- Use dynamic mesh adaption for both steady-state and transient flows.
- Calculate a transient solution using the second-order implicit transient formulation and the density-based implicit solver.
- Create an animation of the transient flow using ANSYS Fluent's transient solution animation feature.

## 6.2. Prerequisites

This tutorial is written with the assumption that you have completed one or more of the introductory tutorials found in this manual:

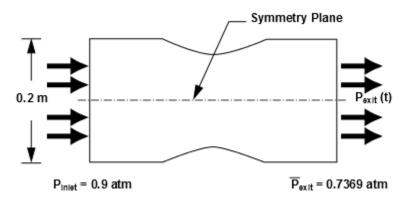
- Introduction to Using ANSYS Fluent in ANSYS Workbench: Fluid Flow and Heat Transfer in a Mixing Elbow (p. 1)
- Parametric Analysis in ANSYS Workbench Using ANSYS Fluent (p. 73)
- Introduction to Using ANSYS Fluent: Fluid Flow and Heat Transfer in a Mixing Elbow (p. 121)

and that you are familiar with the ANSYS Fluent tree and ribbon structure. Some steps in the setup and solution procedure will not be shown explicitly.

# 6.3. Problem Description

The geometry to be considered in this tutorial is shown in Figure 6.1: Problem Specification (p. 268). Flow through a simple nozzle is simulated as a 2D planar model. The nozzle has an inlet height of 0.2 m, and the nozzle contours have a sinusoidal shape that produces a 20% reduction in flow area. Symmetry allows only half of the nozzle to be modeled.

### Figure 6.1: Problem Specification



# 6.4. Setup and Solution

The following sections describe the setup and solution steps for this tutorial:

- 6.4.1. Preparation
- 6.4.2. Reading and Checking the Mesh
- 6.4.3. Solver and Analysis Type
- 6.4.4. Models
- 6.4.5. Materials
- 6.4.6. Operating Conditions
- 6.4.7. Boundary Conditions
- 6.4.8. Solution: Steady Flow

6.4.9. Enabling Time Dependence and Setting Transient Conditions

- 6.4.10. Specifying Solution Parameters for Transient Flow and Solving
- 6.4.11. Saving and Postprocessing Time-Dependent Data Sets

# 6.4.1. Preparation

To prepare for running this tutorial:

- 1. Set up a working folder on the computer you will be using.
- 2. Go to the ANSYS Customer Portal, https://support.ansys.com/training.

### Note

If you do not have a login, you can request one by clicking **Customer Registration** on the log in page.

- 3. Enter the name of this tutorial into the search bar.
- 4. Narrow the results by using the filter on the left side of the page.

- a. Click ANSYS Fluent under Product.
- b. Click **18.0** under **Version**.
- 5. Select this tutorial from the list.
- 6. Click the **unsteady\_compressible\_R180.zip** link to download the input files.
- 7. Unzip the unsteady\_compressible\_R180.zip file you downloaded to your working folder.

The files nozzle.msh and pexit.c can be found in the unsteady\_compressible folder created after unzipping the file.

8. Use Fluent Launcher to start the **2D** version of ANSYS Fluent.

Fluent Launcher displays your **Display Options** preferences from the previous session.

For more information about the Fluent Launcher, see starting ANSYS Fluent using the Fluent Launcher in the Fluent Getting Started Guide.

- 9. Ensure that the **Display Mesh After Reading** and **Workbench Color Scheme** options are enabled.
- 10. Ensure that **Serial** is selected under **Processing Options**.
- 11. Disable the **Double Precision** option.

## 6.4.2. Reading and Checking the Mesh

1. Read the mesh file nozzle.msh.



The mesh for the half of the geometry is displayed in the graphics window.

2. Check the mesh.



ANSYS Fluent will perform various checks on the mesh and will report the progress in the console. Ensure that the reported minimum volume is a positive number.

3. Verify that the mesh size is correct.



💶 Scale N	/lesh			
Domain Ext	tents			 Scaling
Xmin (m)	-0.1	Xmax (m)		<ul> <li>Convert Units</li> <li>Specify Scaling Factors</li> </ul>
Ymin (m)	0	Ymax (m)	0.1	Mesh Was Created In
View Lengt	h Unit In v			Scaling Factors X 1 Y 1 Scale Unscale
		C	lose Help	

Close the Scale Mesh dialog box.

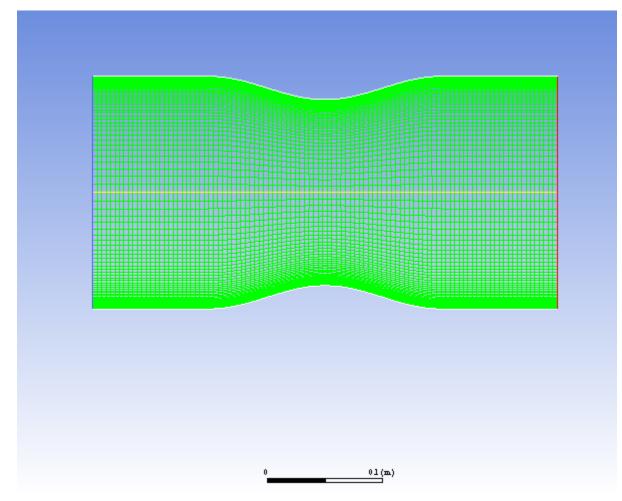
4. Mirror the mesh across the centerline (Figure 6.2: 2D Nozzle Mesh Display with Mirroring (p. 271)).

### $\blacksquare Viewing \rightarrow Display \rightarrow Views...$

<b>E</b> Views							
Views	Actions	Mirror Planes [1/1]					
back front	Default Auto Scale Previous	symmetry					
	Save Delete						
	Read Write	Define Plane Periodic Repeats					
Save Name view-0	Witten	Define					
Apply Camera Close Help							

- a. Select symmetry in the Mirror Planes selection list.
- b. Click **Apply** to refresh the display.
- c. Close the **Views** dialog box.



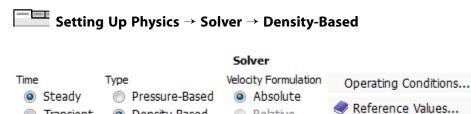


# 6.4.3. Solver and Analysis Type

Oensity-Based

1. Select the solver settings.

Transient



Relative

### a. In the Solver group of the Setting Up Physics tab, select Density-Based from the Type list.

The density-based implicit solver is the solver of choice for compressible, transonic flows without significant regions of low-speed flow. In cases with significant low-speed flow regions, the pressurebased solver is preferred. Also, for transient cases with traveling shocks, the density-based explicit solver with explicit time stepping may be the most efficient.

Planar

b. Retain the default selection of **Steady** from the **Time** list.

### Note

You will solve for the steady flow through the nozzle initially. In later steps, you will use these initial results as a starting point for a transient calculation.

2. For convenience, change the unit of measurement for pressure.

The pressure for this problem is specified in atm, which is not the default unit in ANSYS Fluent. You must redefine the pressure unit as atm.

10000 10000 10	Setting	Up	Domain	$\rightarrow$	Mesh $\rightarrow$ Unit	s
----------------	---------	----	--------	---------------	-------------------------	---

soot-formation-constant-unit	Quantities molec-wt moment number-density particles-conc particles-rate percentage power pressure pressure-gradient resistance site-density soot-formation-constant-unit	-	Units pascal atm psi torr lb/ft2 inches-water Factor 101325 Offset 0	•	Set All to default si british cgs
------------------------------	--	---	--	---	---

a. Select pressure in the Quantities selection list.

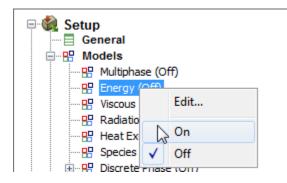
Scroll down the list to find **pressure**.

- b. Select atm in the Units selection list.
- c. Close the Set Units dialog box.

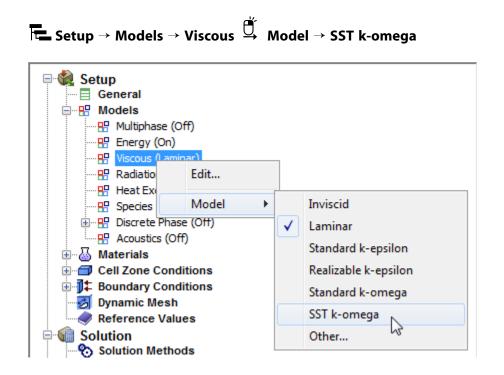
## 6.4.4. Models

1. Enable the energy equation.





2. Select the k-omega SST turbulence model.



## 6.4.5. Materials

1. Define the settings for air, the default fluid material.



Create/Edit Materials					×
Name		Material Type			Order Materials by
air		fluid		-	Name     Chemical Formula
Chemical Formula		FLUENT Fluid M	aterials		
		air		-	FLUENT Database
		Mixture			User-Defined Database
		none			·
Properties				_	
Density (kg/m3)	ideal-gas		▼ Edit	2 ń	
Cp (Specific Heat) (j/kg-k)	constant		- Edit		
	1006.43		•][[[]]	E	
Thermal Conductivity (w/m-k)	constant		▼ Edit		
	0.0242				
Viscosity (kg/m-s)	constant		▼ Edit		
	1.789 <del>4e</del> -05				
(	Change/Create	Delete	Close	Help	

a. Select **ideal-gas** from the **Density** drop-down list in the **Properties** group box, so that the ideal gas law is used to calculate density.

#### Note

ANSYS Fluent automatically enables the solution of the energy equation when the ideal gas law is used, in case you did not already enable it manually in the **Energy** dialog box.

- b. Retain the default values for all other properties.
- c. Click the **Change/Create** button to save your change.
- d. Close the Create/Edit Materials dialog box.

## 6.4.6. Operating Conditions

1. Define the operating pressure.

Setting Up Physics  $\rightarrow$  Solver  $\rightarrow$  Operating Conditions...

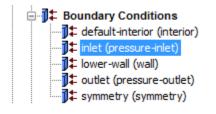
Operating Conditions	<b>-</b>
Pressure	Gravity
Operating Pressure (atm) 0	Gravity
Reference Pressure Location	
X (m) 0	
Y (m) 0	
Z (m) 0	
OK Cancel Help	,

- a. Enter 0 atm for **Operating Pressure**.
- b. Click **OK** to close the **Operating Conditions** dialog box.

Since you have set the operating pressure to zero, you will specify the boundary condition inputs for pressure in terms of absolute pressures when you define them in the next step. Boundary condition inputs for pressure should always be relative to the value used for operating pressure.

## 6.4.7. Boundary Conditions

1. Define the boundary conditions for the nozzle inlet (inlet).





	💶 Pressure Inle	t							×
	Zone Name								
	inlet								
	Momentum	Thermal	Radiation	Species	DPM	Mult	iphase	Potential	UDS
		Reference	e Frame Abso	lute					•
	Ga	uge Total Pre	essure (atm)	0.9			constar	nt	-
	Supersonic/Initial Gauge Pressure (atm) 0.7369 constant						•		
	Direction Specification Method Normal to Boundary								
	r r	Turbulence							
	S	pecification M	lethod Inten	sity and Visc	osity Rat	io			-
			1	Turbulent Int	ensity (	%) 1.5			P
Turbulent Viscosity Ratio 10						P			
			OK	Cancel	Help				

- a. Enter 0.9 atm for Gauge Total Pressure.
- b. Enter 0.7369 atm for Supersonic/Initial Gauge Pressure.

The inlet static pressure estimate is the mean pressure at the nozzle exit. This value will be used during the solution initialization phase to provide a guess for the nozzle velocity.

- c. Retain **Intensity and Viscosity Ratio** from the **Specification Method** drop-down list in the **Turbulence** group box.
- d. Enter 1.5% for **Turbulent Intensity**.
- e. Retain the setting of 10 for **Turbulent Viscosity Ratio**.
- f. Click **OK** to close the **Pressure Inlet** dialog box.
- 2. Define the boundary conditions for the nozzle exit (outlet).

**E** Setup  $\rightarrow$  Boundary Conditions  $\rightarrow$  outlet (pressure-outlet)  $\stackrel{\frown}{\sqcup}$  Edit...

Pressure Outlet
Zone Name
outlet
Momentum         Thermal         Radiation         Species         DPM         Multiphase         Potential         UDS
Backflow Reference Frame Absolute
Gauge Pressure (atm) 0.7369 constant
Backflow Direction Specification Method Normal to Boundary
Average Pressure Specification
Target Mass Flow Rate
Turbulence
Specification Method Intensity and Viscosity Ratio
Backflow Turbulent Intensity (%) 1.5
Backflow Turbulent Viscosity Ratio 10
Acoustic Wave Model
Off
Non Reflecting
OK Cancel Help

- a. Enter 0.7369 atm for Gauge Pressure.
- b. Retain **Intensity and Viscosity Ratio** from the **Specification Method** drop-down list in the **Turbulence** group box.
- c. Enter 1.5% for **Backflow Turbulent Intensity**.
- d. Retain the setting of 10 for Backflow Turbulent Viscosity Ratio.

If substantial backflow occurs at the outlet, you may need to adjust the backflow values to levels close to the actual exit conditions.

e. Click OK to close the Pressure Outlet dialog box.

## 6.4.8. Solution: Steady Flow

In this step, you will generate a steady-state flow solution that will be used as an initial condition for the time-dependent solution.

1. Define the solution parameters.



### Solution Methods

Implicit 👻	
Flux Type	
Roe-FDS 👻	
Spatial Discretization	_
Gradient	1
Least Squares Cell Based 👻	
Flow	
Second Order Upwind 🗸	
Turbulent Kinetic Energy	
Second Order Upwind 🗸	
Specific Dissipation Rate	
Second Order Upwind 🗸	
Transient Formulation	-
▼	
Non-Iterative Time Advancement Frozen Flux Formulation Pseudo Transient High Order Term Relaxation Options	
Convergence Acceleration For Stretched Meshes	
Default	

- a. Retain the default selection of **Least Squares Cell Based** from the **Gradient** drop-down list in the **Spatial Discretization** group box.
- b. Select Second Order Upwind from the Turbulent Kinetic Energy and Specific Dissipation Rate dropdown lists.

Second-order discretization provides optimum accuracy.

2. Modify the Courant Number.

**Solving**  $\rightarrow$  Controls  $\rightarrow$  Controls...

Solution Controls	
Courant Number	
Under-Relaxation Factors	
Turbulent Kinetic Energy	*
0.8	
Specific Dissipation Rate	
0.8	
Turbulent Viscosity	
1	
Solid	
1	
Default	
Equations Limits Advanced	
Help	

a. Enter 50 for the Courant Number.

### Note

The default Courant number for the density-based implicit formulation is 5. For relatively simple problems, setting the Courant number to 10, 20, 100, or even higher value may be suitable and produce fast and stable convergence. However, if you encounter convergence difficulties at the startup of the simulation of a properly set up problem, then you should consider setting the Courant number to its default value of 5. As the solution progresses, you can start to gradually increase the Courant number until the final convergence is reached.

- b. Retain the default values for the Under-Relaxation Factors.
- 3. Enable the plotting of residuals.



Residual Monitors			<b>—</b>		
Options  Print to Console  Plot  Window  1  Curves  Axes  Iterations to Plot  1000	continuity       x-velocity       y-velocity	onitor			
Iterations to Store	Residual Values Normalize Scale Compute Local Sc	Iterations	Convergence Criterion		
OK Plot Renormalize Cancel Help					

- a. Ensure that **Plot** is enabled in the **Options** group box.
- b. Select none from the Convergence Criterion drop-down list.
- c. Click **OK** to close the **Residual Monitors** dialog box.
- 4. Create the surface report definition for mass flow rate at the flow exit.

Solving  $\rightarrow$  Reports  $\rightarrow$  Definitions  $\rightarrow$  New  $\rightarrow$  Surface Report  $\rightarrow$  Mass Flow Rate...

Surface Report Definition		<b>—</b>
Name	Report Type	
mass_flowrate_out	Mass Flow Rate	•
Options	Custom Vectors Vectors of	
Per Surface		<b>v</b>
Average Over	Custom Vectors	
	Field Variable	
Report Files [0/0]	Pressure	•
	Static Pressure	
	Surfaces Filter Text	
	default-interior	
Report Plots [0/0]		
	outlet	
	symmetry	
Create		
Report File		
Report Plot		
Frequency 1		
Print to Console		
Create Output Parameter	New Surface 💌	
	OK Compute Cancel Help	

- a. Enter mass\_flowrate\_out for Name.
- b. Select **outlet** in the **Surfaces** selection list.
- c. In the Create group box, enable Report File, Report Plot and Print to Console.

## Note

When **Report File** is enabled in the **Surface Report Definition** dialog box, the mass flow rate history will be written to a file. If you do not enable this option, the history information will be lost when you exit ANSYS Fluent.

d. Click OK to close the Surface Report Definition dialog box.

mass\_flowrate\_out-rplot and mass\_flowrate\_out-rfile are automatically generated by Fluent and appear in the tree (under Solution/Monitors/Report Plots and Solution/Monitors/Report Files, respectively).

e. Modify the output file name.

Edit Report File				×
Name mass_flowrate_out-rfile				
mass_nownace_out-me				
Available Report Definitions [0/0]			Selected Report Definitions [0/1]	
			mass_flowrate_out	
		Add>>		
		< <remove< td=""><td></td><td></td></remove<>		
Output File Base Name			New V Edit	
noz_ss.out	Browse		New • Eur	
Full File Name				
Get Data Every				
1 💿 iteration	•			
V Prin	t to Console			
	O	Cancel H	elp	

Solution  $\rightarrow$  Monitors  $\rightarrow$  Report Files  $\rightarrow$  mass\_flowrate\_out-rfile  $\stackrel{\bigcirc}{\rightarrow}$  Edit...

- i. Enter noz\_ss.out for Output File Base Name.
- ii. Click **OK** to close the **Edit Report File** dialog box.
- 5. Save the case file (noz\_ss.cas.gz).

**File** 
$$\rightarrow$$
 Write  $\rightarrow$  Case...

6. Initialize the solution.

Solving → Initialization							
	Initializa	ation					
Method		Patch					
O Hybrid	More Settings	Reset Statistics					
Standard	Options	Reset DPM	t = 0 Initialize				

- a. Keep the **Method** at the default of **Hybrid**.
- b. Click Initialize.
- 7. Set up gradient adaption for dynamic mesh refinement.

You will enable dynamic adaption so that the solver periodically refines the mesh in the vicinity of the shocks as the iterations progress. The shocks are identified by their large pressure gradients.

Here 19996	Setting	Up Do	omain →	Adapt	→ Mark/Adapt	Cells $\rightarrow$	Gradient

Cradient Adaption				<b>X</b>
Options           Image: Constant state           Image: Constant state <t< td=""><td>Method Curvature Gradient Iso-Value</td><td>Gradients of Pressure Static Pressure</td><td></td><td><b>•</b></td></t<>	Method Curvature Gradient Iso-Value	Gradients of Pressure Static Pressure		<b>•</b>
Contours Manage Controls	Normalization Standard Scale Normalize Dynamic Interval 100	Min Coarsen Threshold 0.3	Max 0 Refine Threshold 0.7	]
Adap	Mark	Compute Apply	Close Help	

a. Select **Gradient** from the **Method** group box.

The mesh adaption criterion can either be the gradient or the curvature (second gradient). Because strong shocks occur inside the nozzle, the gradient is used as the adaption criterion.

b. Select Scale from the Normalization group box.

Mesh adaption can be controlled by the raw (or standard) value of the gradient, the scaled value (by its average in the domain), or the normalized value (by its maximum in the domain). For dynamic mesh adaption, it is recommended that you use either the scaled or normalized value because the raw values will probably change strongly during the computation, which would necessitate a read-justment of the coarsen and refine thresholds. In this case, the scaled gradient is used.

- c. Enable **Dynamic** in the **Dynamic** group box.
- d. Enter 100 for the Interval.

For steady-state flows, it is sufficient to only seldomly adapt the mesh—in this case an interval of 100 iterations is chosen. For time-dependent flows, a considerably smaller interval must be used.

- e. Retain the default selection of Pressure... and Static Pressure from the Gradients of drop-down lists.
- f. Enter 0.3 for Coarsen Threshold.

g. Enter 0.7 for Refine Threshold.

As the refined regions of the mesh get larger, the coarsen and refine thresholds should get smaller. A coarsen threshold of 0.3 and a refine threshold of 0.7 result in a "medium" to "strong" mesh refinement in combination with the scaled gradient.

- h. Click **Apply** to store the information.
- i. Click the Controls... button to open the Mesh Adaption Controls dialog box.

💶 Mesh Adap	tion Controls	
Options Refine	Zones Filter Text	Min Cell Volume (m3)
Coarsen	fluid	Min # of Cells
		0
		Max # of Cells
		20000 🖨
		Max Level of Refine
		2 🜩
		Volume Weight
		1
	OK Cancel Help	

- i. Retain the default selection of **fluid** in the **Zones** selection list.
- ii. Enter 20000 for Max # of Cells.

To restrict the mesh adaption, the maximum number of cells can be limited. If this limit is violated during the adaption, the coarsen and refine thresholds are adjusted to respect the maximum number of cells. Additional restrictions can be placed on the minimum cell volume, minimum number of cells, and maximum level of refinement.

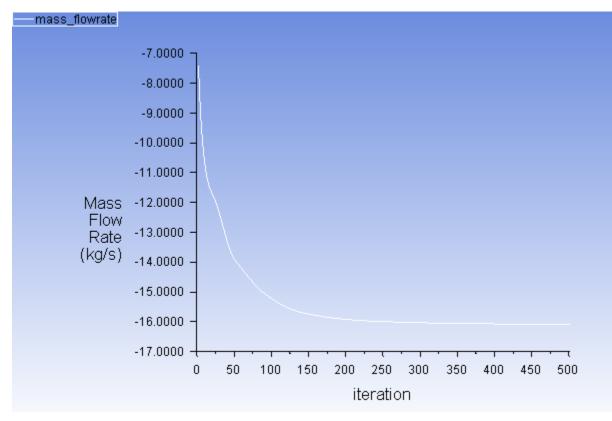
- iii. Click **OK** to save your settings and close the **Mesh Adaption Controls** dialog box.
- j. Click Close to close the Gradient Adaption dialog box.
- 8. Start the calculation by requesting 500 iterations.

Solving  $\rightarrow$  Run Calculation  $\rightarrow$  Advanced...

Run Calculation	
Check Case	Preview Mesh Motion
Number of Iterations	Reporting Interval
Profile Update Interval	
Solution Steering	
Data File Quantities	Acoustic Signals
Calculate	
Help	

- a. Enter 500 for Number of Iterations.
- b. Click **Calculate** to start the steady flow simulation.





9. Save the case and data files (noz\_ss.cas.gz and noz\_ss.dat.gz).

# File $\rightarrow$ Write $\rightarrow$ Case & Data...

### Note

When you write the case and data files at the same time, it does not matter whether you specify the file name with a .cas or .dat extension, as both will be saved.

- 10. Click **OK** in the **Question** dialog box to overwrite the existing file.
- 11. Review a mesh that resulted from the dynamic adaption performed during the computation.

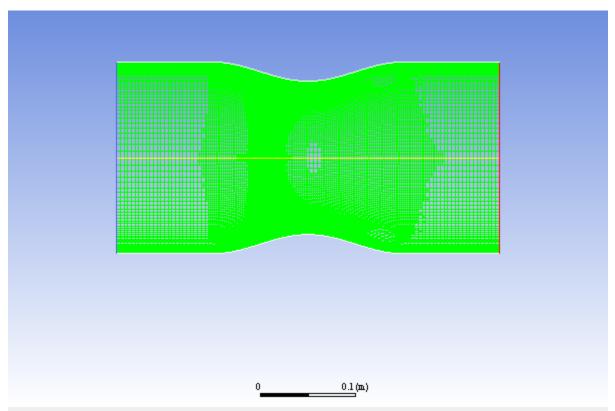
Results $\rightarrow$ Graphics $\rightarrow$ Mesh	🖞 Edit
---	--------

🖸 Mesh Display	/	
Options Nodes	Edge Type All	Surfaces Filter Text
Edges	Feature	default-interior
Faces	Outline	inlet
Partitions		lower-wall
Overset		outlet
Shrink Factor	Feature Angle	symmetry
0	20	
Outline	Interior	
Adjacency		New Surface 💌
	Di	isplay Colors Close Help

- a. Ensure that only the **Edges** option is enabled in the **Options** group box.
- b. Select Feature from the Edge Type list.
- c. Ensure that all of the items are selected from the **Surfaces** selection list.
- d. Click **Display** and close the **Mesh Display** dialog box.

The mesh after adaption is displayed in the graphics window (Figure 6.4: 2D Nozzle Mesh after Adaption (p. 287))





e. Zoom in using the middle mouse button to view aspects of your mesh.

Notice that the cells in the regions of high pressure gradients have been refined.

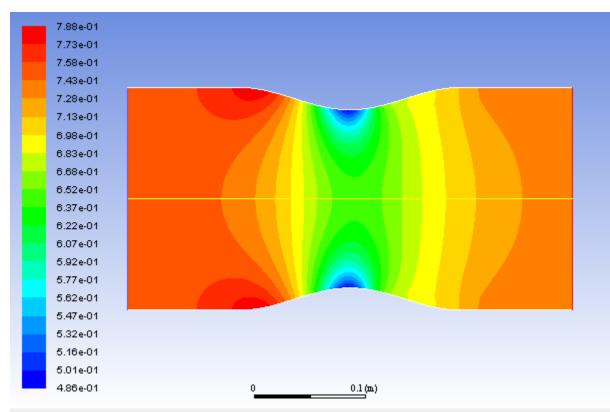
12. Display the steady flow contours of static pressure (Figure 6.5: Contours of Static Pressure (Steady Flow) (p. 288)).

Postprocessing  $\rightarrow$  Graphics  $\rightarrow$  Contours  $\rightarrow$  Edit...

Contours	
Options Filled Node Values	Contours of Pressure  Static Pressure
Global Range Auto Range Clip to Range	Min Max 0 0
<ul><li>Draw Profiles</li><li>Draw Mesh</li></ul>	Surfaces Filter Text
Coloring Banded Smooth	inlet lower-wall outlet symmetry
Levels Setup	New Surface 🔻
	Display Compute Close Help

- a. Enable **Filled** in the **Options** group box.
- b. Click **Display** and close the **Contours** dialog box.

Figure 6.5: Contours of Static Pressure (Steady Flow)



The steady flow prediction in Figure 6.5: Contours of Static Pressure (Steady Flow) (p. 288) shows the expected pressure distribution, with low pressure near the nozzle throat.

13. Display the steady-flow velocity vectors (Figure 6.6: Velocity Vectors Showing Recirculation (Steady Flow) (p. 290)).

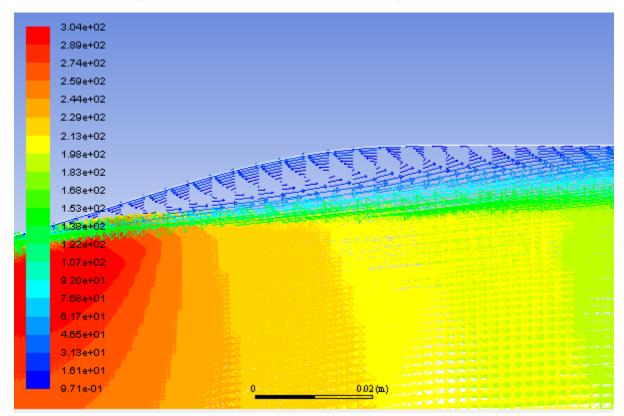
( <b></b>	
Vectors	×.
Options	Vectors of
🗹 Global Range	Velocity
Auto Range	Color by
Clip to Range	Velocity 🔻
Auto Scale Draw Mesh	Velocity Magnitude 🔹
	Min Max
Style	0 0
arrow	
Scale Skip	Surfaces Filter Text
50 0 🚖	default-interior
Vector Options	inlet
Custom Vectors	lower-wall
Custom vectors	outlet
	symmetry
	New Surface 💌
	Display Compute Close Help

## Postprocessing $\rightarrow$ Graphics $\rightarrow$ Vectors $\rightarrow$ Edit...

- a. Enter 50 under Scale.
- b. Click **Display** and close the **Vectors** dialog box.

The steady flow prediction shows the expected form, with a peak velocity of approximately 300 m/s through the nozzle.

You can zoom in on the wall in the expansion region of the nozzle to view the recirculation of the flow as shown in Figure 6.6: Velocity Vectors Showing Recirculation (Steady Flow) (p. 290).



### Figure 6.6: Velocity Vectors Showing Recirculation (Steady Flow)

14. Check the mass flux balance.

### Important

Although the mass flow rate history indicates that the solution is converged, you should also check the mass flux throughout the domain to ensure that mass is being conserved.



E Flux Reports		
Options Mass Flow Rate	Boundaries Filter Text	Results
Total Heat Transfer Rate     Dediction Heat Transfer Rate	default-interior	16 10331 537033010
Radiation Heat Transfer Rate	inlet lower-wall	16.10331527833818
	outlet	-16.10365265607839
	symmetry	
	4	Net Results (kg/s)
Save Output Parameter		-0.0003373777
	Compute Write Close Help	

- a. Retain the default selection of Mass Flow Rate.
- b. Select inlet and outlet in the Boundaries selection list.
- c. Click **Compute** and examine the values displayed in the dialog box.

#### Important

The net mass imbalance should be a small fraction (for example, 0.1%) of the total flux through the system. The imbalance is displayed in the lower right field under **Net Results**. If a significant imbalance occurs, you should decrease your residual tolerances by at least an order of magnitude and continue iterating.

d. Close the **Flux Reports** dialog box.

## 6.4.9. Enabling Time Dependence and Setting Transient Conditions

In this step you will define a transient flow by specifying a transient pressure condition for the nozzle.

1. Enable a time-dependent flow calculation.

Setting Up Physics → Solver → Transient

2. Read the user-defined function (pexit.c), in preparation for defining the transient condition for the nozzle exit.

The pressure at the outlet is defined as a wave-shaped profile, and is described by the following equation:

$$p_{exit}(t) = 0.12\sin(\omega t) + \overline{p}_{exit}$$
(6.1)

where

 $\omega$  = circular frequency of transient pressure (rad/s)

 $\overline{p}_{exit}$  = mean exit pressure (atm)

In this case,  $\omega$ =2200 rad/s, and  $\overline{p}_{e_{xit}}$ = 0.7369 atm.

A user-defined function (pexit.c) has been written to define the equation (Equation 6.1 (p. 291)) required for the pressure profile.

### Note

To input the value of Equation 6.1 (p. 291) in the correct units, the function pexit.c has to be written in SI units.

More details about user-defined functions can be found in the Fluent Customization Manual.

Interpreted UDFs	<b>—</b> ×
Source File Name	
pexit.c	Browse
CPP Command Name	
срр	
Stack Size	
Display Assembly Listing	
Use Contributed CPP	
Interpret Close	Help

## $\blacksquare$ User Defined $\rightarrow$ User Defined $\rightarrow$ Functions $\rightarrow$ Interpreted...

a. Enter pexit.c for Source File Name.

If the UDF source file is not in your working directory, then you must enter the entire directory path for **Source File Name** instead of just entering the file name.

b. Click Interpret.

The user-defined function has already been defined, but it must be compiled within ANSYS Fluent before it can be used in the solver.

- c. Close the Interpreted UDFs dialog box.
- 3. Define the transient boundary conditions at the nozzle exit (**outlet**).



Pressure Outlet	<b>—</b>
Zone Name	
outlet	
Momentum Thermal Radiation Species	DPM Multiphase Potential UDS
Backflow Reference Frame Absolute	•
Gauge Pressure (atm)	udf transient_pressure 🔻
Backflow Direction Specification Method Normal to B	Boundary 🔹
Average Pressure Specification	
Target Mass Flow Rate	
Turbulence	
Specification Method Intensity and	d Viscosity Ratio
Backflow Turbule	nt Intensity (%) 1.5
Backflow Turbuler	nt Viscosity Ratio 10
Acoustic Wave Model	
<ul> <li>Off</li> <li>Non Reflecting</li> </ul>	
OK Cance	I Help

a. Select udf transient\_pressure (the user-defined function) from the Gauge Pressure drop-down list.

b. Click **OK** to close the **Pressure Outlet** dialog box.

4. Update the gradient adaption parameters for the transient case.

**Setting Up Domain**  $\rightarrow$  Adapt  $\rightarrow$  Mark/Adapt Cells  $\rightarrow$  Gradient...

a. Enter 10 for Interval in the Dynamic group box.

For the transient case, the mesh adaption will be done every 10 time steps.

- b. Enter 0.3 for Coarsen Threshold.
- c. Enter 0.7 for Refine Threshold.

The refine and coarsen thresholds have been changed during the steady-state computation to meet the limit of 20000 cells. Therefore, you must reset these parameters to their original values.

- d. Click Apply to store the values.
- e. Click Controls... to open the Mesh Adaption Controls dialog box.
  - i. Enter 8000 for **Min # of Cells**.
  - ii. Enter 30000 for Max # of Cells.

You must increase the maximum number of cells to try to avoid readjustment of the coarsen and refine thresholds. Additionally, you must limit the minimum number of cells to 8000, because you should not have a coarse mesh during the computation (the current mesh has approximately 20000 cells).

4

- iii. Click OK to close the Mesh Adaption Controls dialog box.
- f. Close the Gradient Adaption dialog box.

# 6.4.10. Specifying Solution Parameters for Transient Flow and Solving

1. Modify the **mass\_flowrate\_out-rfile** report file definition.

<b>F</b> Solution $\rightarrow$ Monitors $\rightarrow$ Report Files $\rightarrow$ mass_flowrate_out-rfile $\xrightarrow{\Box}$ Edit
---

Edit Report File		
Name		
mass_flowrate_out-rfile		
Available Report Definitions [0/3]	Selected Report Definitions [0/1]	
delta-time flow-time iters-per-timestep	Add>> < <remove< td=""></remove<>	
Output File Base Name noz_uns.out Browse Full File Name :\solution_files\\noz_ss.out Get Data Every 1 🔶 time-step  V Print to Console	New  Edit	
OK Cancel Help		

- a. Enter noz\_uns.out for Output File Base Name.
- b. Select time-step from the Get Data Every drop-down list.
- c. Click **OK** to close the **Edit Report File** dialog box.
- 2. Modify the mass\_flowrate\_out-rplot report plot definition.

Solution  $\rightarrow$  Monitors  $\rightarrow$  Report Plots  $\rightarrow$  mass\_flowrate\_out-rplot  $\stackrel{\Box}{\rightarrow}$  Edit...

Edit Report Plot	
Name	
mass_flowrate_out-rplot	
Available Report Definitions [0/0]	Selected Report Definitions [0/1]
	mass_flowrate_out
	Add>>
	< <remove< td=""></remove<>
Options	
Plot Window	New - Edit
2 • Curves Axes	
Get Data Every	
1 🔷 time-step 🔻	
Plot Title mass_flowrate_out-rplot	
X-Axis Label time-step 💌	
Y-Axis Label Mass Flow Rate	
Print to Console	
0	Cancel Help

a. For Get Data Every, retain the value of 1 and select time-step from the drop-down list.

Because each time step requires 10 iterations, a smoother plot will be generated by plotting at every time step.

- b. Select time-step from the X-Axis Label drop-down list.
- c. Click OK to close the Edit Report File dialog box.
- 3. Save the transient solution case file (noz\_uns.cas.gz).

**File**  $\rightarrow$  Write  $\rightarrow$  Case...

4. Modify the plotting of residuals.



- a. Ensure that **Plot** is enabled in the **Options** group box.
- b. Ensure none is selected from the Convergence Criterion drop-down list.
- c. Set the **Iterations to Plot** to 100.
- d. Click **OK** to close the **Residual Monitors** dialog box.
- 5. Define the time step parameters.

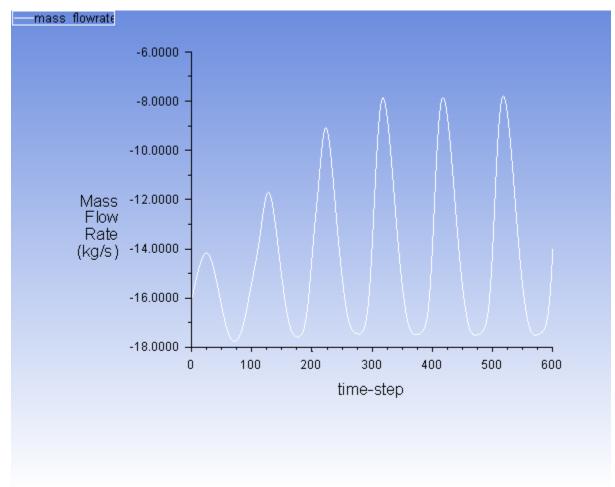
The selection of the time step is critical for accurate time-dependent flow predictions. Using a time step of 2.85596 x  $10^{-5}$  seconds, 100 time steps are required for one pressure cycle. The pressure cycle begins and ends with the initial pressure at the nozzle exit.

Check Case	Preview Mesh Motion.	
ime Stepping Method	Time Step Size (s)	
Fixed 🔻	2.85596e-5	
Settings	Number of Time Steps	
	600 ≑	
Options		
Extrapolate Variable	5	
Data Sampling for T	ime Statistics	
Sampling Interval		
1	Sampling Options	
Time Sample	ed (s) 0	
Solid Time Step		
<ul> <li>User Specified</li> <li>Automatic</li> </ul>		
I Automatic		
ax Iterations/Time Ste	1	
rofile Update Interval		
Data File Quantities	Acoustic Signals	

- a. Enter 2.85596e-5 s for Time Step Size.
- b. Enter 600 for Number of Time Steps.
- c. Enter 10 for Max Iterations/Time Step.
- d. Click Calculate to start the transient simulation.

By requesting 600 time steps, you are asking ANSYS Fluent to compute six pressure cycles. The mass flow rate history is shown in Figure 6.7: Mass Flow Rate History (Transient Flow) (p. 297).





- 6. Optionally, you can review the effect of dynamic mesh adaption performed during transient flow computation as you did in steady-state flow case.
- 7. Save the transient case and data files (noz\_uns.cas.gz and noz\_uns.dat.gz).

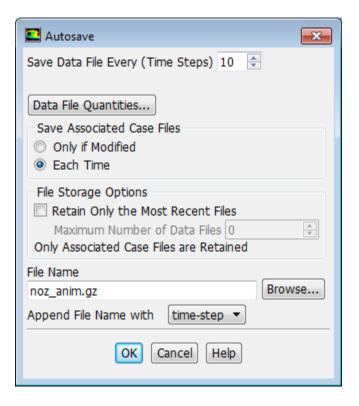
File → Write → Case & Data...

# 6.4.11. Saving and Postprocessing Time-Dependent Data Sets

At this point, the solution has reached a time-periodic state. To study how the flow changes within a single pressure cycle, you will now continue the solution for 100 more time steps. You will use ANSYS Fluent's solution animation feature to save contour plots of pressure and Mach number at each time step, and the autosave feature to save case and data files every 10 time steps. After the calculation is complete, you will use the solution animation playback feature to view the animated pressure and Mach number plots over time.

1. Request the saving of case and data files every 10 time steps.





- a. Enter 10 for **Save Data File Every**.
- b. Select Each Time for Save Associated Case Files.
- c. Retain the default selection of time-step from the Append File Name with drop-down list.
- d. Enter noz\_anim.gz for File Name.

When ANSYS Fluent saves a file, it will append the time step value to the file name prefix ( noz\_anim). The standard extensions (.cas and .dat) will also be appended. By adding the optional extension .gz to the end of the file name, you instruct ANSYS Fluent to save the case and data files in compressed format. This will yield file names of the form noz\_anim-1-00640.cas.gz and noz\_anim-1-00640.dat.gz, where 00640 is the time step number.

e. Click OK to close the Autosave dialog box.

## Tip

If you have constraints on disk space, you can restrict the number of files saved by ANSYS Fluent by enabling the **Retain Only the Most Recent Files** option and setting the **Maximum Number of Data Files** to a nonzero number.

2. Create an animation definition for the nozzle pressure contour plot.

Solving  $\rightarrow$  Activities  $\rightarrow$  Create  $\rightarrow$  Solution Animations...

Animation Definition	
Name: pressure	
Record after every	1 🗧 Time Step 🔻
Storage Type	In Memory
Storage Directory	sible/solution_files
Window Id	3
Animation Object	=
residuals	
contour-1 surf-mon-1-pset	
sun-mon-1-pset	
New Object   Edit Object	
Save Close Help	

- a. Enter pressure for the Name.
- b. Select Time Step for Record after every.

The default value of 1 in the integer number entry box instructs ANSYS Fluent to update the animation sequence at every time step.

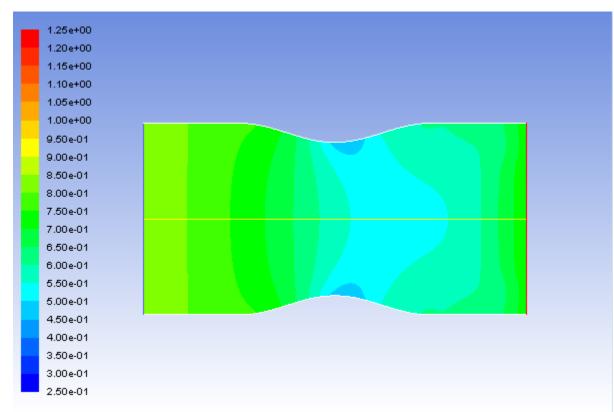
c. Select In Memory from the Storage Type drop-down.

The **In Memory** option is acceptable for a small 2D case such as this. For larger 2D or 3D cases, saving animation files with either the **Metafile** or **PPM Image** option is preferable, to avoid using too much of your machine's memory.

- d. Enter **3** for the **Window Id**.
- e. Click New Object and select Contours... from the drop-down list to open the associated dialog box.

Contours			<b>—</b> ×
Contour Name			
contour-1			
Options	Contours of		
✓ Filled	Pressure		
Node Values	Static Pressure 🔹		•
Global Range Auto Range	Min (atm)	Max (atm)	
Clip to Range	0.25	1.25	
Draw Profiles	Surfaces Filter Text		x
	default-interior inlet		
Coloring	iniet lower-wall		
Banded	outlet		
Smooth	symmetry		
Colormap Options	zone-surface-5		
colonnap options	New Surface 🔻		
Save/Display Compute Close Help			

- i. Select **Pressure...** and **Static Pressure** from the **Contours of** drop-down lists.
- ii. Ensure that **Filled** is enabled in the **Options** group box.
- iii. Disable Auto Range.
- iv. Enter 0.25 atm for Min and 1.25 atm for Max.
- v. Click Save/Display and close the Contours dialog box.



## vi. Figure 6.8: Pressure Contours at t=0.017136 s

- f. Ensure contour-1 is selected in the Animation Object group box.
- g. Click Save and close the Animation Definition dialog box.
- 3. Create an animation definition for the Mach number contour plot.

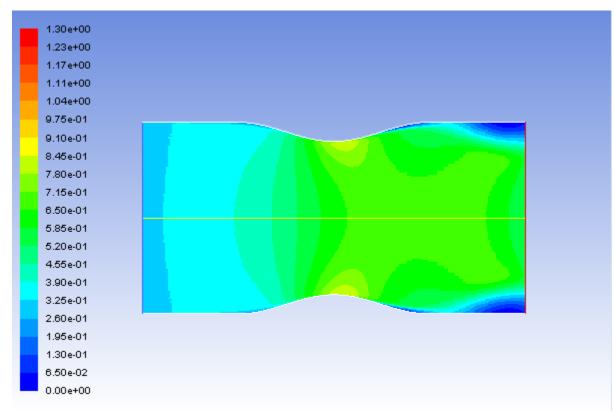
**Solving**  $\rightarrow$  Activities  $\rightarrow$  Create  $\rightarrow$  Solution Animations...

Animation Definition		
Name: mach-number		
Record after every	1 🗧 Time Step 🔻	
Storage Type	In Memory	
Storage Directory	sible/solution_files	
Window Id	4	
Animation Object	=	
residuals contour-1		
contour-2		
surf-mon-1-pset		
New Object 🔹 Edit Object		
Save Close Help		

- a. Enter **mach-number** for the **Name**.
- b. Select Time Step for Record after every.

The default value of 1 in the integer number entry box instructs ANSYS Fluent to update the animation sequence at every time step.

- c. Ensure that In Memory is selected from the Storage Type drop-down.
- d. Enter 4 for the Window Id.
- e. Click New Object and select Contours... from the drop-down list to open the associated dialog box.
  - i. Select Velocity... and Mach Number from the Contours of drop-down lists.
  - ii. Ensure that **Filled** is enabled in the **Options** group box.
  - iii. Disable Auto Range.
  - iv. Enter 0.00 for Min and 1.30 for Max.
  - v. Click Save/Display and close the Contours dialog box.



## vi. Figure 6.9: Mach Number Contours at t=0.017136 s

- f. Ensure contour-2 is selected in the Animation Object group box.
- g. Click Save and close the Animation Definition dialog box.
- 4. Continue the calculation by requesting 100 time steps.

By requesting 100 time steps, you will march the solution through an additional 0.0028 seconds, or roughly one pressure cycle.

With the autosave and animation features active (as defined previously), the case and data files will be saved approximately every 0.00028 seconds of the solution time; animation files will be saved every 0.000028 seconds of the solution time.



Run Calculation	
Check Case	Preview Mesh Motion
Time Stepping Method	Time Step Size (s)
Fixed 💌	2.85596e-05
Settings	Number of Time Steps
	100
Options	
Extrapolate Variables	Statistics
Sampling Interval	
	Sampling Options
Time Sampled (s	0
Max Iterations/Time Step	Reporting Interval
Profile Update Interval	
1	
Data File Quantities	Acoustic Signals
Calculate	
Help	

Enter 100 for Number of Time Steps and click Calculate.

When the calculation finishes, you will have ten pairs of case and data files and there will be 100 pairs of contour plots stored in memory. In the next few steps, you will play back the animation sequences and examine the results at several time steps after reading in pairs of newly saved case and data files.

5. Play the animation of the pressure contours.



Playback	×	
Playback Playback Mode Play Once Start Frame Increment End Frame 1  1 1 1 1 1 1 1 1 1 1 1 1 1 1 1 1 1 1	Animation Sequences Sequences pressure mach-number	
Slow Replay Speed Fast	Delete Delete All	
Write/Record Format Animation Frames	Picture Options	
Write Read Close Help		

a. Retain the default selection of pressure in the Sequences selection list.

Ensure that tab window 4 is open in the graphics window.

- b. Click the play button (the second from the right in the group of buttons in the **Playback** group box).
- c. Close the **Playback** dialog box.

Examples of pressure contours at t=0.017993 s (the 630th time step) and t=0.019135 s (the 670th time step) are shown in Figure 6.10: Pressure Contours at t=0.017993 s(p. 306) and Figure 6.11: Pressure Contours at t=0.019135 s (p. 306).

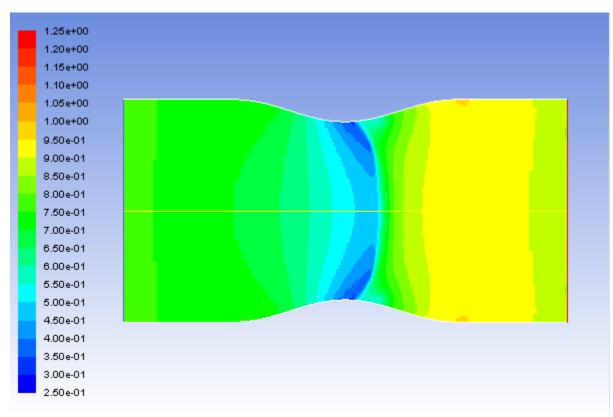
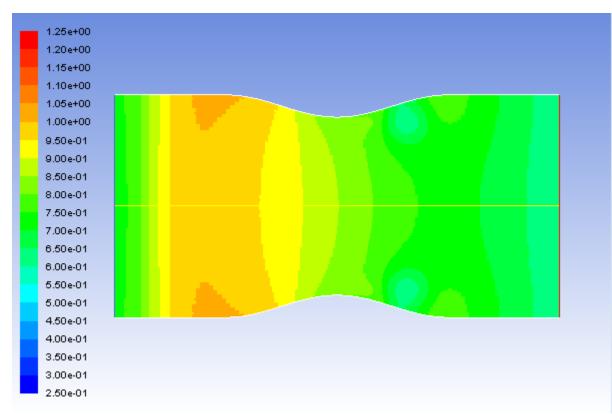


Figure 6.10: Pressure Contours at t=0.017993 s





6. In a similar manner to steps 4 and 5, select the appropriate active window and animation sequence name for the Mach number contours.

Examples of Mach number contours at t=0.017993 s and t=0.019135 s are shown in Figure 6.12: Mach Number Contours at t=0.017993 s (p. 307) and Figure 6.13: Mach Number Contours at t=0.019135 s(p. 308).

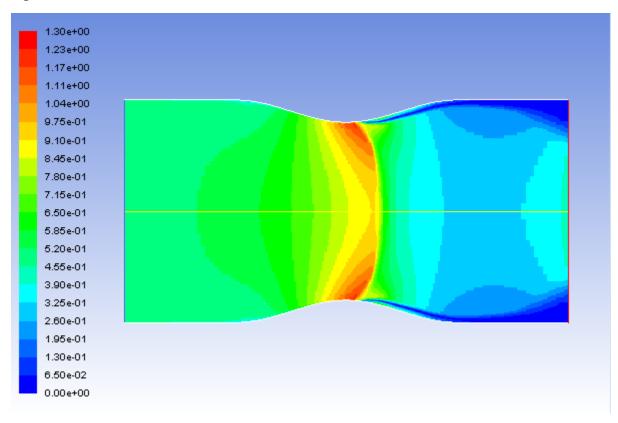
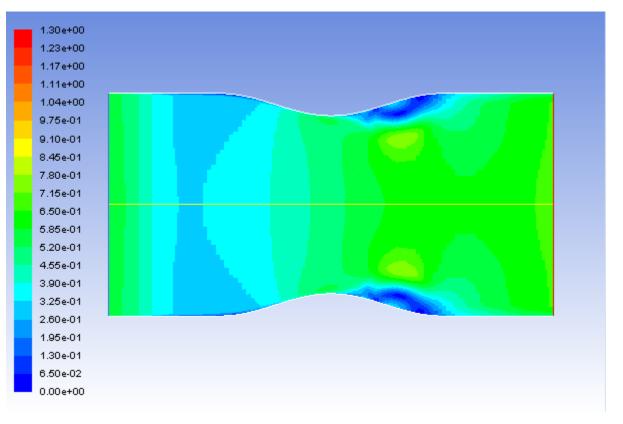


Figure 6.12: Mach Number Contours at t=0.017993 s



### Figure 6.13: Mach Number Contours at t=0.019135 s

### Тір

ANSYS Fluent gives you the option of exporting an animation as an MPEG file or as a series of files in any of the hardcopy formats available in the **Save Picture** dialog box (including TIFF and PostScript).

To save an MPEG file, select **MPEG** from the **Write/Record Format** drop-down list in the **Playback** dialog box and then click the **Write** button. The MPEG file will be saved in your working folder. You can view the MPEG movie using an MPEG player (for example, Windows Media Player or another MPEG movie player).

To save a series of TIFF, PostScript, or other hardcopy files, select **Picture Frames** in the **Write/Record Format** drop-down list in the **Playback** dialog box. Click the **Picture Op-tions...** button to open the **Save Picture** dialog box and set the appropriate parameters for saving the hardcopy files. Click **Apply** in the **Save Picture** dialog box to save your modified settings. Click **Save...** to select a directory in which to save the files. In the **Playback** dialog box, click the **Write** button. ANSYS Fluent will replay the animation, saving each frame to a separate file in your working folder.

If you want to view the solution animation in a later ANSYS Fluent session, you can select **Animation Frames** as the **Write/Record Format** and click **Write**.

### Warning

Because the solution animation was stored in memory, it will be lost if you exit ANSYS Fluent without saving it in one of the formats described previously. Note that only the animation-frame format can be read back into the **Playback** dialog box for display in a later ANSYS Fluent session.

- 7. Read the case and data files for the 660th time step (**noz\_anim-1-00660.cas.gz** and **noz\_an-im-1-00660.dat.gz**) into ANSYS Fluent.
- 8. Plot vectors at t=0.018849 s (Figure 6.14: Velocity Vectors at t=0.018849 s (p. 310)).

<b>Vectors</b>	×
Options Global Range	Vectors of Velocity
<ul> <li>Auto Range</li> <li>Clip to Range</li> </ul>	Color by Velocity
<ul> <li>Auto Scale</li> <li>Draw Mesh</li> </ul>	Velocity Magnitude <ul> <li>Min (m/s)</li> <li>Max (m/s)</li> </ul> <ul> <li>Max (m/s)</li> </ul>
Style arrow	0.3684946 262.8595
Scale Skip	Surfaces Filter Text
50 50 ÷	default-interior inlet lower-wall outlet symmetry
	New Surface   Display Compute Close Help
	رغبي ( <u>معمد</u> ) المعاد (

Postprocessing  $\rightarrow$  Graphics  $\rightarrow$  Vectors  $\rightarrow$  Edit...

- a. Ensure Auto Scale is enabled under Options.
- b. Enter 50 under Scale.
- c. Enter 50 under Skip.
- d. Click **Display** and close the **Vectors** dialog box.

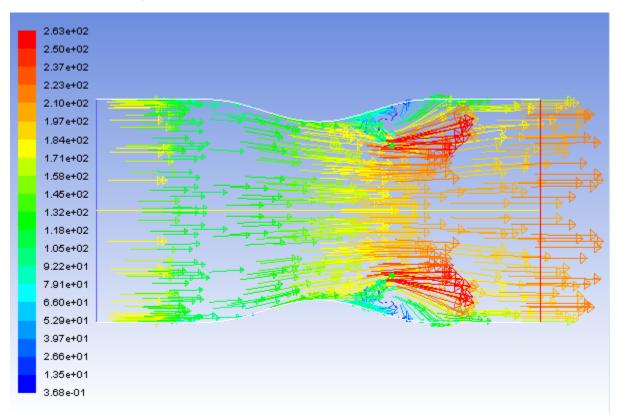


Figure 6.14: Velocity Vectors at t=0.018849 s

The transient flow prediction in Figure 6.14: Velocity Vectors at t=0.018849 s (p. 310) shows the expected form, with peak velocity of approximately 260 m/s through the nozzle at t=0.018849 seconds.

9. In a similar manner to steps 7 and 8, read the case and data files saved for other time steps of interest and display the vectors.

# 6.5. Summary

In this tutorial, you modeled the transient flow of air through a nozzle. In doing so, you learned how to:

- generate a steady-state solution as an initial condition for the transient case.
- set solution parameters for implicit time-stepping and apply a user-defined transient pressure profile at the outlet.
- use mesh adaption to refine the mesh in areas with high pressure gradients to better capture the shocks.
- automatically save solution information as the transient calculation proceeds.
- create and view solution animations of the transient flow.

# **6.6. Further Improvements**

This tutorial guides you through the steps to generate a second-order solution. You may be able to increase the accuracy of the solution even further by using an appropriate higher-order discretization scheme and by adapting the mesh further. Mesh adaption can also ensure that the solution is independ-

ent of the mesh. These steps are demonstrated in Introduction to Using ANSYS Fluent: Fluid Flow and Heat Transfer in a Mixing Elbow (p. 121).

# **Chapter 7: Modeling Radiation and Natural Convection**

This tutorial is divided into the following sections:

- 7.1. Introduction
- 7.2. Prerequisites
- 7.3. Problem Description
- 7.4. Setup and Solution
- 7.5. Summary
- 7.6. Further Improvements

# 7.1. Introduction

In this tutorial, combined radiation and natural convection are solved in a three-dimensional square box on a mesh consisting of hexahedral elements.

This tutorial demonstrates how to do the following:

- Use the surface-to-surface (S2S) radiation model in ANSYS Fluent.
- Set the boundary conditions for a heat transfer problem involving natural convection and radiation.
- Calculate a solution using the pressure-based solver.
- Display velocity vectors and contours of wall temperature, surface cluster ID, and radiation heat flux.

# 7.2. Prerequisites

This tutorial is written with the assumption that you have completed one or more of the introductory tutorials found in this manual:

- Introduction to Using ANSYS Fluent in ANSYS Workbench: Fluid Flow and Heat Transfer in a Mixing Elbow (p. 1)
- Parametric Analysis in ANSYS Workbench Using ANSYS Fluent (p. 73)
- Introduction to Using ANSYS Fluent: Fluid Flow and Heat Transfer in a Mixing Elbow (p. 121)

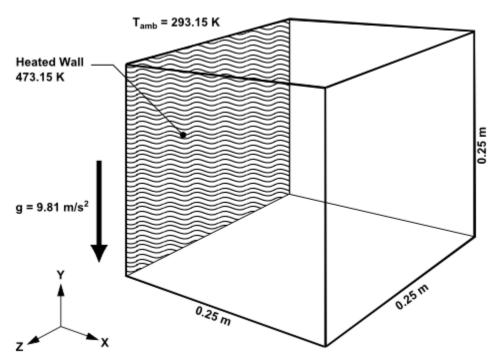
and that you are familiar with the ANSYS Fluent tree and ribbon structure. Some steps in the setup and solution procedure will not be shown explicitly.

# 7.3. Problem Description

The problem to be considered is shown schematically in Figure 7.1: Schematic of the Problem (p. 314). A three-dimensional box  $(0.25m \times 0.25m \times 0.25m)$  has a hot wall of aluminum at 473.15 K. All other walls are made of an insulation material and are subject to radiative and convective heat transfer to the surroundings, which are at 293.15 K. Gravity acts downwards. The medium contained in the box is assumed not to emit, absorb, or scatter radiation. All walls are gray. The objective is to compute the

flow and temperature patterns in the box, as well as the wall heat flux, using the surface-to-surface (S2S) model available in ANSYS Fluent.

The working fluid has a Prandtl number of approximately 0.71, and the Rayleigh number based on L (0.25) is  $1 \times 10^8$ . This means the flow is most likely laminar. The Planck number  $k / (4\sigma LT_0^3)$  is 0.006, and measures the relative importance of conduction to radiation.





# 7.4. Setup and Solution

The following sections describe the setup and solution steps for this tutorial:

- 7.4.1. Preparation
- 7.4.2. Reading and Checking the Mesh
- 7.4.3. Solver and Analysis Type
- 7.4.4. Models
- 7.4.5. Defining the Materials
- 7.4.6. Operating Conditions
- 7.4.7. Boundary Conditions
- 7.4.8. Obtaining the Solution
- 7.4.9. Postprocessing
- 7.4.10. Comparing the Contour Plots after Varying Radiating Surfaces
- 7.4.11. S2S Definition, Solution, and Postprocessing with Partial Enclosure

# 7.4.1. Preparation

To prepare for running this tutorial:

1. Set up a working folder on the computer you will be using.

2. Go to the ANSYS Customer Portal, https://support.ansys.com/training.

### Note

If you do not have a login, you can request one by clicking **Customer Registration** on the log in page.

- 3. Enter the name of this tutorial into the search bar.
- 4. Narrow the results by using the filter on the left side of the page.
  - a. Click ANSYS Fluent under Product.
  - b. Click **18.0** under **Version**.
- 5. Select this tutorial from the list.
- 6. Click the radiation\_natural\_convection\_R180.zip link to download the input files.
- 7. Unzip radiation\_natural\_convection\_R180.zip to your working folder.

The mesh file rad.msh.gz can be found in the radiation\_natural\_convection folder created after unzipping the file.

8. Use Fluent Launcher to start the **3D**, single precision (disable **Double Precision**) version of ANSYS Fluent.

Fluent Launcher displays your **Display Options** preferences from the previous session.

For more information about Fluent Launcher, see starting ANSYS Fluent using the Fluent Launcher in the Fluent Getting Started Guide.

- 9. Ensure that the **Display Mesh After Reading** and **Workbench Color Scheme** options are enabled.
- 10. Ensure that Serial is selected under Processing Options.

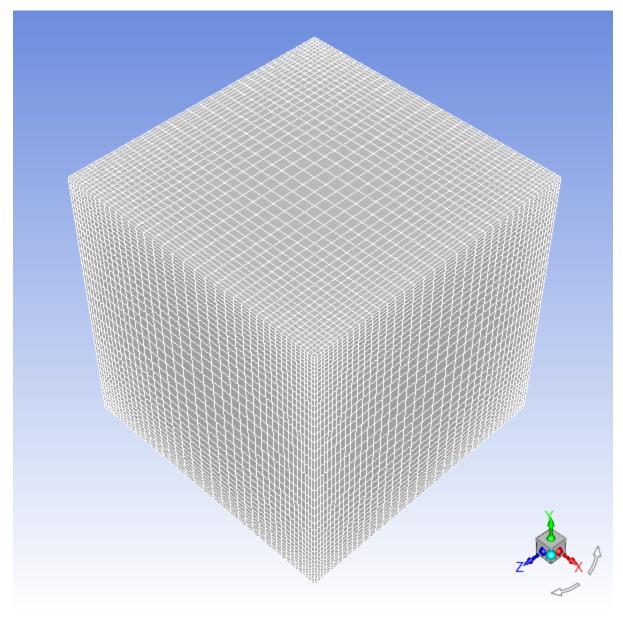
## 7.4.2. Reading and Checking the Mesh

1. Read the mesh file rad.msh.gz.

### File → Read → Mesh...

As the mesh is read, messages will appear in the console reporting the progress of the reading and the mesh statistics. The mesh size will be reported as 64,000 cells. Once reading is complete, the mesh will be displayed in the graphics window.





2. Check the mesh.

### **Setting Up Domain** $\rightarrow$ Mesh $\rightarrow$ Check

ANSYS Fluent will perform various checks on the mesh and report the progress in the console. Make sure that the reported minimum volume is a positive number.

# 7.4.3. Solver and Analysis Type

1. Confirm the solver settings and enable gravity.

Setup → ♀General

General
Mesh
Scale Check Report Quality
Display
Solver
Type Velocity Formulation
Pressure-Based Absolute
O Density-Based
Time Steady Transient
Gravity Units
Gravitational Acceleration
X (m/s2) 0
Y (m/s2) -9.81
Z (m/s2) 0
Help

- a. Retain the default settings of pressure-based steady-state solver in the **Solver** group box.
- b. Enable the **Gravity** option.
- c. Enter  $-9.81 \text{ m/s}^2$  for **Y** in the **Gravitational Acceleration** group box.

## 7.4.4. Models

1. Enable the energy equation.

Setup → Mod	lels → Energy 🗳 Or
	se (Off)
	Edit
Badiatic Bates Bates Bates	
- Species	

2. Set up the **Surface to Surface (S2S)** radiation model.

Radiation Model	
Model Off Rosseland P1 Discrete Transfer (DTRM) Surface to Surface (S2S) Discrete Ordinates (DO) Monte Carlo (MC)	Iteration Parameters Energy Iterations per Radiation Iteration 10 Maximum Number of Radiation Iterations 5 Residual Convergence Criteria 0.001 View Factors and Clustering Settings Compute/Write/Read Read Existing File
Solar Load Model Off Solar Ray Tracing DO Irradiation Solar Calculator	OK Cancel Help

# **T** Setup $\rightarrow$ Models $\rightarrow$ Radiation $\stackrel{\bigcirc}{\rightarrow}$ Model $\rightarrow$ Surface to Surface (S2S)

The surface-to-surface (S2S) radiation model can be used to account for the radiation exchange in an enclosure of gray-diffuse surfaces. The energy exchange between two surfaces depends in part on their size, separation distance, and orientation. These parameters are accounted for by a geometric function called a "view factor".

The S2S model assumes that all surfaces are gray and diffuse. Thus according to the gray-body model, if a certain amount of radiation is incident on a surface, then a fraction is reflected, a fraction is absorbed, and a fraction is transmitted. The main assumption of the S2S model is that any absorption, emission, or scattering of radiation by the medium can be ignored. Therefore only "surface-to-surface" radiation is considered for analysis.

For most applications the surfaces in question are opaque to thermal radiation (in the infrared spectrum), so the surfaces can be considered opaque. For gray, diffuse, and opaque surfaces it is valid to assume that the emissivity is equal to the absorptivity and that reflectivity is equal to 1 minus the emissivity.

When the S2S model is used, you also have the option to define a "partial enclosure". This option allows you to disable the view factor calculation for walls with negligible emission/absorption or walls that have uniform temperature. The main advantage of this option is to speed up the view factor calculation and the radiosity calculation.

### a. Click the Settings... button to open the View Factors and Clustering dialog box.

You will define the view factor and cluster parameters.

View Factors and Clusterir	ıg	<b>—</b>			
Clustering Options Manual Automatic					
Manual Faces per Surface Cluster for Flow Boundary Zones 1 Apply to All Walls	Automatic Maximum Faces p Surface Cluster 10 Compute	⊅er ∳			
View Factors Basis Face to Face Cluster to Cluster		Method Ray Tracing Hemicube			
Surfaces Blocking Nonblocking		Parameters Resolution 10 🖨 Subdivisions 5 🔄 Normalized Separation 5 Distance			
Zones Participating inView	Zones Participating inView Factor Calculation Select				
OK Cancel Help					

- i. Retain the value of 1 for Faces per Surface Cluster for Flow Boundary Zones in the Manual group box.
- ii. Click Apply to All Walls.

The S2S radiation model is computationally very expensive when there are a large number of radiating surfaces. The number of radiating surfaces is reduced by clustering surfaces into surface "clusters". The surface clusters are made by starting from a face and adding its neighbors and their neighbors until a specified number of faces per surface cluster is collected.

For a small problem, the default value of 1 for **Faces per Surface Cluster for Flow Boundary Zones** is acceptable. For a large problem you can increase this number to reduce the memory requirement for the view factor file that is saved in a later step. This may also lead to some reduction in the computational expense. However, this is at the cost of some accuracy. This tutorial illustrates the influence of clusters.

- iii. Ensure Ray Tracing is selected from the Method list in the View Factors group box.
- iv. Click OK to close the View Factors and Clustering dialog box.
- b. Click the **Compute/Write/Read...** button in the **View Factors and Clustering** group box to open the **Select File** dialog box and to compute the view factors.

The file created in this step will store the cluster and view factor parameters.

- i. Enter **rad\_1.s2s.gz** as the file name for **S2S File**.
- ii. Click **OK** in the **Select File** dialog box.

#### Note

The size of the view factor file can be very large if not compressed. It is highly recommended to compress the view factor file by providing .gz or .Z extension after the name (that is, rad\_1.gz or rad\_1.Z). For small files, you can provide the .s2s extension after the name.

ANSYS Fluent will print an informational message describing the progress of the view factor calculation in the console.

c. Click OK to close the Radiation Model dialog box.

# 7.4.5. Defining the Materials

1. Set the properties for air.

Create/Edit Materials				
ame	Material Type			Order Materials by
r		fluid		Name
hemical Formula		Fluent Fluid Materials		Chemical Formula
	air		•	Fluent Database
	Mbcture			User-Defined Database.
	none		Ŧ	
roperties				
Density (kg/m3) ideal-	gas	▼ Edit		
Cp (Specific Heat) (j/kg-k) const	ant	▼ Edit		
1021				
	+	▼ Edt		
Thermal Conductivity (w/m-k) const		Edt		
0.037	'1			
Viscosity (kg/m-s) const	ant	<ul> <li>Edit</li> </ul>		
2.485	ie-05			
Molecular Weight (kg/kmol) const	ant	▼ Edit		
28.96				
20.74	v .			

- a. Select ideal-gas from the Density drop-down list.
- b. Enter 1021 J/kg-K for **Cp (Specific Heat)**.

- c. Enter 0.0371 W/m-K for Thermal Conductivity.
- d. Enter 2.485e-05 kg/m-s for Viscosity.
- e. Retain the default value of 28.966 kg/kmol for Molecular Weight.
- f. Click Change/Create and close the Create/Edit Materials dialog box.
- 2. Define the new material, **insulation**.

<b>T</b> Setup $\rightarrow$ Materials $\rightarrow$ Solid $\rightarrow$ alum	ninum 🗓	Edit
---	---------	------

Create/Edit Materials		<b>—</b>
Name	Material Type	Order Materials by
insulation	solid	<ul> <li>Name</li> </ul>
Chemical Formula	Fluent Solid Materials	Chemical Formula
	insulation	Fluent Database
	Mixture	
	none	User-Defined Database
Properties		
Density (kg/m3)	constant 🔹 Edit	
	50	
Cp (Specific Heat) (j/kg-k)	constant   Edit	
	800	
Thermal Conductivity (w/m-k)	constant   Edit	
	0.09	
	Change/Create Delete Close Help	

- a. Enter insulation for Name.
- b. Delete the entry in the **Chemical Formula** field.
- c. Enter 50 kg/m<sup>3</sup> for **Density**.
- d. Enter 800 J/kg-K for Cp (Specific Heat).
- e. Enter 0.09 W/m-K for Thermal Conductivity.
- f. Click Change/Create.
- g. Click **No** when the **Question** dialog box appears, asking if you want to overwrite aluminum.

The **Create/Edit Materials** dialog box will be updated to show the new material, **insulation**, in the **Fluent Solid Materials** drop-down list.

h. Close the Create/Edit Materials dialog box.

## 7.4.6. Operating Conditions

Specify operating density.

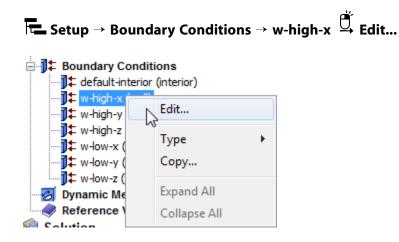
100001 [H	Setting	Up	Phy	sics →	Solver	→ Or	perating	Conditions.	••

Operating Conditions		×	
Pressure Operating Pressure (pascal) 101325 P Reference Pressure Location X (m) 0 P Y (m) 0 P Z (m) 0 P	Gravity	P P	
OK Cancel Help			

- 1. In the **Operating Conditions** dialog box, select the **Specified Operating Density** check box.
- 2. Enter 0 for **Operating Density** and click **OK** to close the **Operating Conditions** dialog box.

## 7.4.7. Boundary Conditions

1. Set the boundary conditions for the front wall (w-high-x).



The Wall dialog box opens.

💶 Wall			
Zone Name			
w-high-x			
Adjacent Cell Zone fluid			
Momentum Thermal	Radiation Species DPM Multiphase	e UDS Wall Film	Potential
Thermal Conditions			
Heat Flux	Heat Transfer Coefficient (w/m2-k)	5	constant 🔹
Temperature	Free Stream Temperature (k)	293.15	constant 🔹
<ul> <li>Convection</li> <li>Radiation</li> </ul>	External Emissivity	0.75	constant 🔻
Mixed	External Radiation Temperature (k)	293.15	constant 🔹
via System Coupling	Internal Emissivity	0.95	constant 🔹
via Mapped Interface		Wall Thickness (m)	0.05 P
	Heat Generation Rate (w/m3)	0	constant 🔹
		Shell Conduction	1 Layer Edit
Material Name insulation	Edit		
	OK Cancel He	ql	

- a. Click the Thermal tab and select Mixed from the Thermal Conditions list.
- b. Select insulation from the Material Name drop-down list.
- c. Enter 5 W/m<sup>2</sup>-K for **Heat Transfer Coefficient**.
- d. Enter 293.15 K for Free Stream Temperature.
- e. Enter 0.75 for External Emissivity.
- f. Enter 293.15 K for External Radiation Temperature.
- g. Enter 0.95 for Internal Emissivity.
- h. Enter 0.05 m for Wall Thickness.
- i. Click **OK** to close the **Wall** dialog box.
- 2. Copy boundary conditions to define the side walls w-high-z and w-low-z.

**E** Setup  $\rightarrow$  Boundary Conditions  $\rightarrow$  w-high-x  $\stackrel{\textcircled{}}{\rightarrow}$  Copy...

Copy Conditions	<b>X</b>				
From Boundary Zone Filter Text	To Boundary Zones Filter Text				
w-high-x	w-high-y				
w-high-y	w-high-z				
w-high-z	w-low-x				
w-low-x	w-low-y				
w-low-y	w-low-z				
w-low-z					
Copy Close Help					
	11.				

- a. Ensure w-high-x is selected in the From Boundary Zone selection list.
- b. Select w-high-z and w-low-z from the To Boundary Zones selection list.
- c. Click Copy.
- d. Click **OK** when the **Question** dialog box opens asking whether you want to copy the boundary conditions of w-high-x to all the selected zones.

17

- e. Close the **Copy Conditions** dialog box.
- 3. Set the boundary conditions for the heated wall (**w-low-x**).

<b>F</b> Setup $\rightarrow$ Boundary Conditions $\rightarrow$ w-low-x $\stackrel{\bigcup}{\rightarrow}$ Edit	E Setup →	Boundary	Conditions $\rightarrow$	w-low-x	Edit
---	-----------	----------	--------------------------	---------	------

💶 Wall									<b>×</b>
Zone Name					_				
w-low-x									
Adjacent Cell Zone					_				
fluid									
Momentum Thermal	Radiation	Species	DPM	Multip	hase	UDS	Wall Film	Potential	
Thermal Conditions									
Heat Flux		1	Temperat	ture (k)	473.15			constant	-
Temperature		Ir	iternal Er	nissivity	0.95			constant	•
<ul> <li>Convection</li> <li>Radiation</li> </ul>					W	/all Thick	ness (m) 0		P
<ul> <li>Mixed</li> </ul>	He	at Generati	on Rate (	(w/m3)	0			constant	-
via System Coupling					Sł	nell Cond	uction 1	ayer	Edit
via Mapped Interface									
Material Name									
aluminum 🔻	Edit								
			ОК Са	ncel	lelp				

- a. Click the Thermal tab and select Temperature from the Thermal Conditions list.
- b. Retain the default selection of aluminum from the Material Name drop-down list.
- c. Enter 473.15 K for Temperature.
- d. Enter 0.95 for Internal Emissivity.
- e. Click OK to close the Wall dialog box.
- 4. Set the boundary conditions for the top wall (w-high-y).

Setup $ ightarrow$ Boundary Conditions $ ightarrow$ w-high	-y 🗳 Edit
--	-----------

🔁 Wall									<b>×</b>
Zone Name									
w-high-y									
Adjacent Cell Zo	one								
fluid									
Momentum	Thermal	Radiation	Species	DPM	Multiphase	UDS	Wall Film	Potential	
Thermal Condi	tions								
Heat Flux		He	at Transfer O	oefficie	nt (w/m2-k)	3		constant	•
<ul> <li>Temperat</li> <li>Convertion</li> </ul>			Free Strea	m Tem	perature (k)	293.15		constant	•
<ul> <li>Convection</li> <li>Radiation</li> </ul>	n	External Emissivity			0.75		constant	•	
Mixed	Mixed External Radiation Temperature (k)		perature (k)	293.15		constant	•		
	n Coupling	Internal Emissivity		nal Emissivity	0.95		constant	•	
🔘 via Mappe	id Interface					Wall	Thickness (m	n) 0.05	P
			Heat Gene	ration R	Rate (w/m3)	0		constant	•
						Shell	Conduction	1 Layer	Edit
Material Name insulation	•	Edit							
			(	ОК	Cancel He	lp .			

- a. Click the Thermal tab and select Mixed from the Thermal Conditions list.
- b. Select insulation from the Material Name drop-down list.
- c. Enter 3 W/m<sup>2</sup>-K for Heat Transfer Coefficient.
- d. Enter 293.15 K for Free Stream Temperature.
- e. Enter 0.75 for External Emissivity.
- f. Enter 293.15 K for External Radiation Temperature.
- g. Enter 0.95 for Internal Emissivity.
- h. Enter 0.05 m for Wall Thickness.
- i. Click **OK** to close the **Wall** dialog box.

5. Copy boundary conditions to define the bottom wall (**w-low-y**) as previously done in this tutorial.

<b>Setup</b> $\rightarrow$ <b>Boundary Conditions</b> $\rightarrow$ w-high-y	🖞 Сору
--	--------

Copy Conditions	
From Boundary Zone Filter Text	To Boundary Zones Filter Text
w-high-x w-high-y w-high-z w-low-x w-low-y w-low-z	w-high-x w-high-z w-low-x <mark>w-low-y</mark> w-low-z
Сору	Close Help

- a. Ensure w-high-y is selected in the From Boundary Zone selection list.
- b. Select **w-low-y** from the **To Boundary Zones** selection list.
- c. Click **Copy**.
- d. Click **OK** when the **Question** dialog box opens asking whether you want to copy the boundary conditions of w-high-y to all the selected zones.
- e. Close the Copy Conditions dialog box.

## 7.4.8. Obtaining the Solution

1. Set the solution parameters.



Solution Methods
Pressure-Velocity Coupling
Scheme
Coupled 👻
Spatial Discretization
Gradient
Least Squares Cell Based 🔻
Pressure
Body Force Weighted
Density
Second Order Upwind 🔻
Momentum
Second Order Upwind 🔻
Energy
Second Order Upwind 🔻
Transient Formulation
· · · · · · · · · · · · · · · · · · ·
Non-Iterative Time Advancement
Frozen Flux Formulation
Pseudo Transient
Warped-Face Gradient Correction
High Order Term Relaxation Options
Default
Help

- a. Select **Coupled** from the **Scheme** drop-down list in the **Pressure-Velocity Coupling** group box.
- b. Select **Body Force Weighted** from the **Pressure** drop-down list in the **Spatial Discretization** group box.
- c. Retain the default selection of **Second Order Upwind** from the **Momentum** and **Energy** drop-down lists.
- d. Enable the Pseudo Transient option.
- 2. Set the convergence criteria for you simulation.



Residual Monitors					×
Options	Equations				
Print to Console	Residual	Monito	r Check Converge	nce Absolute Criteria	
V Plot	continuity	<b>V</b>	<b>V</b>	0.0001	
Window	x-velocity	<b>V</b>	$\mathbf{\nabla}$	0.0001	
1 🗘 Curves Axes	y-velocity			0.0001	
Iterations to Plot	z-velocity		$\checkmark$	0.0001	
1000 🗢	energy		<b>V</b>	1e-07	
Iterations to Store 1000	<ul> <li>Residual Values</li> <li>Normalize</li> <li>Scale</li> <li>Compute Loca</li> </ul>	al Scale	Iterations 5	Convergence Criterion absolute Convergence Conditions	•
ОК	Plot Renorma	alize	ancel Help		

- a. Ensure that **Plot** is enabled in the **Options** group box.
- b. Enter 0.0001 for the Absolute Criteria of continuity, x-, y-, and z-velocity.
- c. Enter 1e-7 for the Absolute Criteria of energy.

Decreasing the criteria for these residuals will improve the accuracy of the solution.

- d. Click OK
- 3. Initialize the solution.

Solving → Initialization



- a. Retain the default selection of Hybrid Initialization from the Initialization Methods list.
- b. Click Initialize.
- 4. Create a surface report definition to aid in judging convergence.

It is good practice to use reports of physical solution quantities together with residual monitors when determining whether a solution is converged. In this step you will create a surface report definition for the average temperature on the z=0 plane.

a. Create the new surface, **zz\_center\_z**.

Setting Up D	Domain $ ightarrow$ Surface $ ightarrow$ Cr	reate → Iso-Surface	
Iso-Surface			<b>—</b> ×-
Surface of Constant Mesh	•	From Surface Filter Text	
	• 1ax (m) 0.125	default-interior w-high-x w-high-y w-high-z w-low-x w-low-y	
New Surface Name zz_center_z	►	From Zones Fiter Text	
	Create	Manage Close Help	

- i. Select Mesh... and Z-Coordinate from the Surface of Constant drop-down lists.
- ii. Click **Compute** and retain the default value of 0 for **Iso-Values**.
- iii. Enter **zz\_center\_z** for **New Surface Name**.

#### Note

If you want to delete or otherwise manipulate any surfaces, click **Manage...** to open the **Surfaces** dialog box.

- iv. Click Create and close the Iso-Surface dialog box.
- b. Create the surface report definition.

**Solving**  $\rightarrow$  Reports  $\rightarrow$  Definitions  $\rightarrow$  New  $\rightarrow$  Surface Report  $\rightarrow$  Area-Weighted Average...

Surface Report Definition	
Name	Report Type
surf-mon-1	Area-Weighted Average
Options	Custom Vectors
	Vectors of
Per Surface	<b></b>
Average Over	Custom Vectors
1	
	Field Variable
Report Files [0/0]	Temperature
	Static Temperature 👻
	Surfaces Filter Text
	default-interior
Report Plots [0/0]	w-high-x
	w-high-y w-high-z
	w-low-x
	w-low-y
	w-low-z
	zz_center_z
Create	
Report File	
Report Plot	
Frequency 1	
Print to Console	Highlight Surfaces
Create Output Parameter	New Surface 🔻
OK Con	npute Cancel Help

- i. Enter **surf-mon-1** for the **Name** of the surface report definition.
- ii. In the Create group box, enable Report Plot and Print to Console.

### Note

Unlike residual values, data from other reports is not saved as part of the solution set when the ANSYS Fluent data file is saved. If you want to access the surface report data in future ANSYS Fluent sessions, you can enable the **Report File** option. The report file will be saved in your working directory.

- iii. Select Temperature... and Static Temperature from the Field Variable drop-down lists.
- iv. Select **zz\_center\_z** from the **Surfaces** selection list.
- v. Click **OK** to save the surface report definition settings and close the **Surface Report Definition** dialog box.

5. Save the case file (rad\_a\_1.cas.gz).



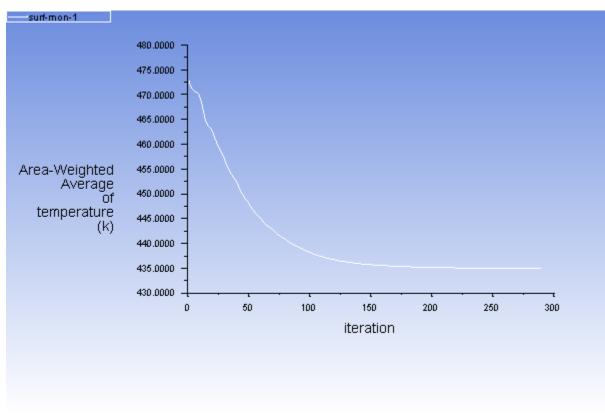
6. Start the calculation by requesting 400 iterations.

Solving → Run Calculation→Advanced	•••
------------------------------------	-----

Run Calculation	
Check Case	Update Dynamic Mesh
-Pseudo Transient Opti	ons
Fluid Time Scale	
Time Step Method	Pseudo Time Step (s)
User Specified	1
O Automatic	
Number of Iterations 400 Profile Update Interval 1	1
1	
Data File Quantities	Acoustic Signals
	Acoustic Signals Acoustic Sources FFT

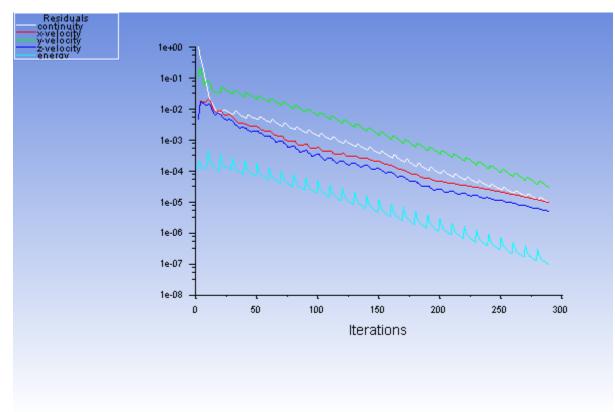
- a. Select User Specified from the Time Step Method list.
- b. Retain the default value of 1 for **Pseudo Time Step**.
- c. Enter 400 for Number of Iterations.
- d. Click Calculate.





The surface report history shows that the average temperature on **zz\_center\_z** has stabilized, thus confirming that the solution has indeed reached convergence. You can view the behavior of the residuals (Figure 7.4: Scaled Residuals (p. 333)) in the **Scaled Residuals** graphics window tab.

Figure 7.4: Scaled Residuals



7. Save the case and data files (rad\_a\_1.cas.gz,rad\_a\_1.dat.gz).

**File**  $\rightarrow$  Write  $\rightarrow$  Case & Data...

# 7.4.9. Postprocessing

- 1. Enable the postprocessing view by clicking *in the objects toolbar and selecting Post Processing*.
- 2. Disable lighting.

**E** Results 
$$\rightarrow$$
 Graphics  $\stackrel{\frown}{\hookrightarrow}$  Lights...

- a. Disable Light On and click Apply.
- b. Close the **Lights** dialog box.
- 3. Create a new surface, **zz\_x\_side**, which will be used later to plot wall temperature.

```
Postprocessing \rightarrow Surface \rightarrow Create \rightarrow Line/Rake...
```

Line/Rake Surface	<b>—</b>		
Options Type Line Tool Reset	Number of Points		
End Points			
x0 (m) -0.125	x1 (m) 0.125		
y0 (m) 0	y1 (m) 0		
z0 (m) 0.125	z1 (m) 0.125		
Select Points with Mouse			
New Surface Name			
zz_x_side			
Create Manage Close Help			

- a. Enter (-0.125, 0, 0.125) for (x0, y0, z0), respectively.
- b. Enter (0.125, 0, 0.125) for (x1, y1, z1), respectively.
- c. Enter **zz\_x\_side** for **New Surface Name**.

### Note

If you want to delete or otherwise manipulate any surfaces, click **Manage...** to open the **Surfaces** dialog box.

- d. Click Create and close the Line/Rake Surface dialog box.
- 4. Display contours of static temperature.



Contours			×
Options	Contours of		
V Filled	Temperature 🔻		
Node Values	Static Temperature		•
Global Range Auto Range	Min	Max	
Clip to Range	421	473.5	
Draw Profiles	Surfaces Filter Text		x
Coloring Banded Smooth Levels Setup 20 1	w-high-y w-high-z w-low-x w-low-y w-low-z zz_center_z zz_x_side New Surface Display Compute	Close Help	

- a. Enable the **Filled** option in the **Options** group box.
- b. Select Temperature... and Static Temperature from the Contours of drop-down lists.
- c. Select **zz\_center\_z** from the **Surfaces** selection list.
- d. Enable the **Draw Mesh** option in the **Options** group box to open the **Mesh Display** dialog box.
  - i. Ensure **Outline** is selected in the **Edge Type** group box.
  - ii. Close the Mesh Display dialog box.
- e. Disable the **Auto Range** option.
- f. Enter 421 K for Min and 473.15 K for Max.
- g. Click **Display**, rotate the view as shown in Figure 7.5: Contours of Static Temperature (p. 336), and close the **Contours** dialog box.

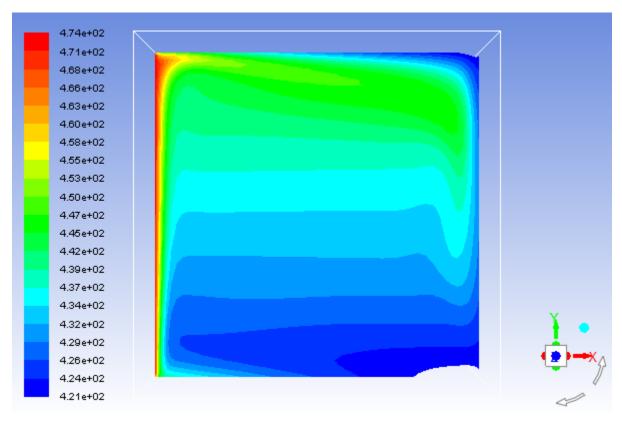


Figure 7.5: Contours of Static Temperature

A regular check for most buoyant cases is to look for evidence of stratification in the temperature field. This is observed as nearly horizontal bands of similar temperature. These may be broken or disturbed by buoyant plumes. For this case you can expect reasonable stratification with some disturbance at the vertical walls where the air is driven around. Inspection of the temperature contours in Figure 7.5: Contours of Static Temperature (p. 336) reveals that the solution appears as expected.

5. Create and display a contour definition for wall temperature (surfaces in contact with the fluid).

Postprocessing  $\rightarrow$  Graphics  $\rightarrow$  Contours  $\rightarrow$  New...

Contours	
Contour Name	
wall-temperature	
Options	Contours of
V Filled	Pressure
Node Values     Clobal Papers	Static Pressure 🔹
<ul> <li>Global Range</li> <li>Auto Range</li> </ul>	Min Max
Clip to Range	0 0
Draw Profiles	Surfaces Filter Text
	default-interior
Coloring	w-high-x
Banded	w-high-y w-high-z
Smooth	w-low-x
	w-low-y
Colormap Options	w-low-z
	zz_center_z
	zz_x_side
	New Surface 🔻
	Save/Display Compute Close Help

- a. Enter wall-temperature for Contour Name.
- b. Ensure that the **Filled** option is enabled in the **Options** group box.
- c. Disable the **Node Values** option.
- d. Select Temperature... and Wall Temperature from the Contours of drop-down lists.
- e. Select all surfaces except **default-interior** and **zz\_x\_side** in the **Surfaces** selection list.
- f. Enter 413 K for Min and 473.15 K for Max.
- g. Click Save/Display, and rotate the view as shown in Figure 7.6: Contours of Wall Temperature (p. 338).

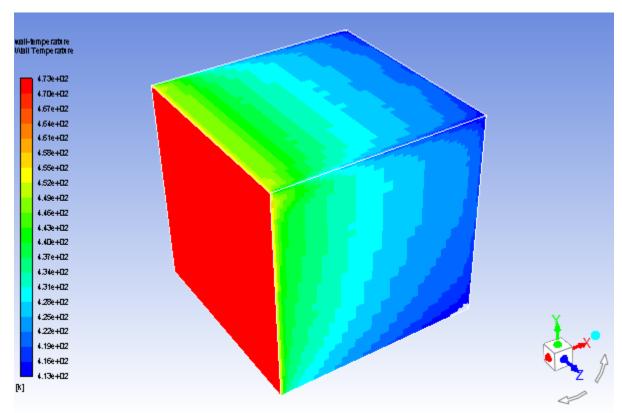


Figure 7.6: Contours of Wall Temperature

6. Display contours of radiation heat flux.



Contours	
Options Filled Node Values Global Range Auto Range Clip to Range Draw Profiles Draw Mesh	Contours of Wall Fluxes
	Radiation Heat Flux
	Min (w/m2) Max (w/m2) -265.2101 889.5444
	Surfaces Filter Text
Coloring Banded Smooth Levels Setup 20 1	default-interior w-high-x w-high-y w-high-z w-low-x w-low-y w-low-z zz_center_z zz_x_side New Surface
	Display Compute Close Help

- a. Ensure that the **Filled** option is enabled in the **Options** group box.
- b. Enable Auto Range.
- c. Select Wall Fluxes... and Radiation Heat Flux from the Contours of drop-down list.
- d. Make sure that all surfaces except **default-interior** and **zz\_x\_side** are selected in the **Surfaces** selection list.
- e. Click Display.
- f. Close the **Contours** dialog box.

*Figure 7.7: Contours of Radiation Heat Flux (p. 340) shows the radiating wall (w-low-x) with positive heat flux and all other walls with negative heat flux.* 

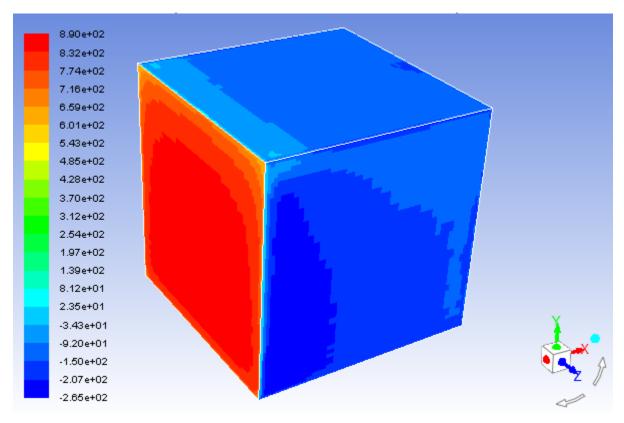


Figure 7.7: Contours of Radiation Heat Flux

7. Display vectors of velocity magnitude.



<b>E</b> Vectors	
Options Global Range Auto Range Clip to Range Auto Scale Draw Mesh	Vectors of Velocity Color by Velocity Velocity Magnitude
Style arrow Scale Skip 7 0 ÷ Vector Options Custom Vectors	Min Max 0 0 Surfaces Filter Text Filter Text Filter Text Filter Text Mefault-interior w-high-x w-high-y w-high-z w-low-x
	w-low-y w-low-z zz_center_z zz_x_side New Surface Display Compute Close Help

- a. Retain the default selection of **Velocity** from the **Vectors of** drop-down list.
- b. Retain the default selection of **Velocity...** and **Velocity Magnitude** from the **Color by** drop-down lists.
- c. Select **zz\_center\_z** from the **Surfaces** selection list.
- d. Enable the Draw Mesh option in the Options group box to open the Mesh Display dialog box.
  - i. Ensure that **Outline** is selected from the **Edge Type** list.
  - ii. Click Display and close the Mesh Display dialog box.
- e. Enter 7 for Scale.
- f. Click **Display** and rotate the view as shown in Figure 7.8: Vectors of Velocity Magnitude (p. 342).
- g. Close the Vectors dialog box.

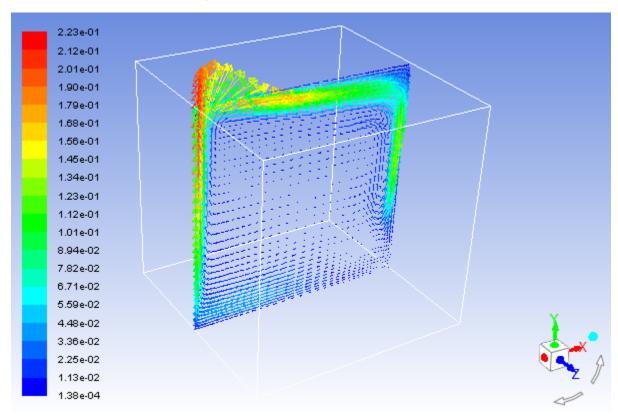


Figure 7.8: Vectors of Velocity Magnitude

8. Compute view factors and radiation emitted from the front wall (w-high-x) to all other walls.

In the Postprocessing tab, click S2S Information... (Model Specific group box).

S2S Information				×
Report Options View Factors Incident Radiation	From Filter Text		To Filter Text	
Boundary Types [0/11]	w-high-z w-low-z w-low-y w-low-z		w-high-y w-high-z w-low-y w-low-y w-low-z	
Compute Write Cose Help				

- a. Ensure that the View Factors option is enabled in the Report Options group box.
- b. Enable the Incident Radiation option.

- c. Select w-high-x from the From selection list.
- d. Select all zones except w-high-x from the To selection list.
- e. Click Compute and close the S2S Information dialog box.

The computed values of the view factors and incident radiation are displayed in the console. A view factor of approximately 0.2 for each wall is a good value for the square box.

9. Compute the total heat transfer rate.

0001 HHH	Postprocessing	→ Reports →	Fluxes

E Flux Reports			×
Options Mass Flow Rate	Boundaries Filter Text	Results	
<ul> <li>Total Heat Transfer Rate</li> <li>Radiation Heat Transfer Rate</li> </ul>	default-interior w-high-x w-high-y	-12.81945238015545 -12.61944048777846	
	w-high-z w-low-x	-13.04612051883683 63.65117862698035	
	w-low-y w-low-z	-12.13333957003079 -13.04567611232134	
	٠	•	Þ
Save Output Parameter		Net Results (w)	_
		-0.01285044	
	Compute Write Close Help		

- a. Select Total Heat Transfer Rate from the Options list.
- b. Select all boundary zones except default-interior from the Boundaries selection list.
- c. Click Compute.

Note

The energy imbalance is approximately 0.004%.

10. Compute the total heat transfer rate for **w-low-x**.



E Flux Reports		<b>—</b>
Options Mass Flow Rate	Boundaries Filter Text	Results
<ul> <li>Total Heat Transfer Rate</li> <li>Radiation Heat Transfer Rate</li> </ul>	default-interior w-high-x w-high-y w-high-z w-low-x w-low-y w-low-z	63.65117862698035
	4	٠
Save Output Parameter		Net Results (w)
		63.65118
	Compute Write Close Help	.H.

- a. Retain the selection of Total Heat Transfer Rate from the Options list.
- b. Deselect all boundary zones and select **w-low-x** from the **Boundaries** selection list.
- c. Click Compute.

#### Note

The net heat load is approximately 63 W.

11. Compute the radiation heat transfer rate.



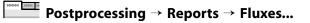
E Flux Reports		<b>×</b>
Options Mass Flow Rate	Boundaries Filter Text	Results
<ul> <li>Total Heat Transfer Rate</li> <li>Radiation Heat Transfer Rate</li> </ul>	default-interior w-high-x w-high-y w-high-z w-low-x w-low-y w-low-y	-10.98288437480712 -6.944987894713171 -10.70552600315407 51.86710101153235 -12.53661540261715 -10.70333174316133
Save Output Parameter	4	<ul> <li>Net Results (w)</li> <li>-0.006244407</li> </ul>
	Compute Write Close Help	ħ.

- a. Select Radiation Heat Transfer Rate from the Options list.
- b. Select all boundary zones except **default-interior** from the **Boundaries** selection list.
- c. Click Compute.

#### Note

The heat imbalance is approximately -0.007 W.

12. Compute the radiation heat transfer rate for **w-low-x**.



E Flux Reports		<b>—</b>
Options Mass Flow Rate	Boundaries Filter Text	Results
<ul> <li>Total Heat Transfer Rate</li> <li>Radiation Heat Transfer Rate</li> </ul>	default-interior w-high-x w-high-y w-high-z w-low-x w-low-y w-low-y w-low-z	51.86710101153235
	٠	٠
Save Output Parameter		Net Results (w)
		51.8671
	Compute Write Close Help	

- a. Retain the selection of Radiation Heat Transfer Rate from the Options list.
- b. Deselect all boundary zones and select **w-low-x** from the **Boundaries** selection list.
- c. Click **Compute** and close the **Flux Reports** dialog box.

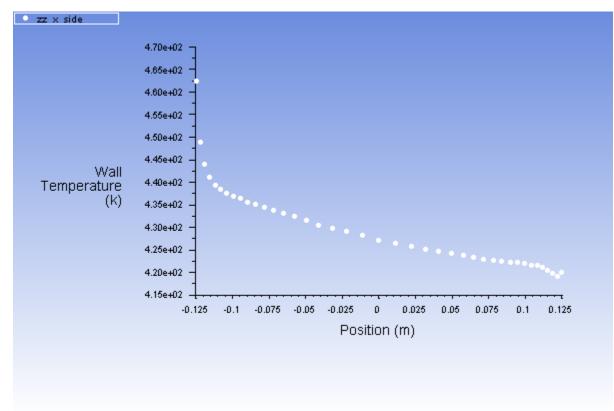
The net heat load is approximately 51 W. After comparing the total heat transfer rate and radiation heat transfer rate, it can be concluded that radiation is the dominant mode of heat transfer.

13. Display the temperature profile for the side wall.

Postprocessing  $\rightarrow$  Plots  $\rightarrow$  XY Plot  $\rightarrow$  Edit...

Solution XY Plot		
Options Vode Values Position on X Axis Position on Y Axis Write to File Order Points File Data	Plot Direction X 1 Y 0 Z 0	Y Axis Function Temperature Wall Temperature X Axis Function Direction Vector Surfaces Filter Text default-interior w-high-x w-high-y w-high-z w-low-x w-low-x w-low-z zz_center_z zz_x_side New Surface
E	Plot Axes Curves	Close Help

- a. Select Temperature... and Wall Temperature from the Y Axis Function drop-down lists.
- b. Retain the default selection of **Direction Vector** from the **X Axis Function** drop-down list.
- c. Select **zz\_x\_side** from the **Surfaces** selection list.
- d. Click Plot (Figure 7.9: Temperature Profile Along the Outer Surface of the Box (p. 348)).
- e. Enable the Write to File option and click the Write... button to open the Select File dialog box.
  - i. Enter tp\_1.xy for XY File.
  - ii. Click **OK** in the **Select File** dialog box.
- f. Disable the Write to File option.
- g. Close the **Solution XY Plot** dialog box.



#### Figure 7.9: Temperature Profile Along the Outer Surface of the Box

14. Save the case and data files (rad\_b\_1.cas.gz and rad\_b\_1.dat.gz).

File  $\rightarrow$  Write  $\rightarrow$  Case & Data...

## 7.4.10. Comparing the Contour Plots after Varying Radiating Surfaces

1. Increase the number of faces per cluster to 10.

**E** Setup 
$$\rightarrow$$
 Models  $\rightarrow$  Radiation  $\stackrel{\text{dis}}{\rightarrow}$  Edit...

- a. Click the Settings... button to open the View Factors and Clustering dialog box.
  - i. Enter 10 for Faces per Surface Cluster for Flow Boundary Zones in the Manual group box.
  - ii. Click Apply to All Walls.
  - iii. Click OK to close the View Factors and Clustering dialog box.
- b. Click the **Compute/Write/Read...** button to open the **Select File** dialog box and to compute the view factors.

Specify a name for the S2S file that will store the cluster and view factor parameters.

- i. Enter rad\_10.s2s.gz for S2S File.
- ii. Click **OK** in the **Select File** dialog box.

- c. Click **OK** to close the **Radiation Model** dialog box.
- 2. Right-click Solution Initialization and select Initialize.



3. Start the calculation by requesting 400 iterations.



The solution will converge in approximately 309 iterations.

4. Save the case and data files (rad\_10.cas.gz and rad\_10.dat.gz).

**File**  $\rightarrow$  Write  $\rightarrow$  Case & Data...

- 5. In a similar manner described in the steps 13.a 13.g of Postprocessing (p. 333), display the temperature profile for the side wall and write it to a file named  $tp_{10.xy}$ .
- 6. Repeat the procedure, outlined in steps 1 5 of this section, for 100, 400, 800, and 1600 faces per surface cluster and save the respective S2S files (for example, rad\_100.s2s.gz), case and data files (for example, rad\_100.cas.gz), and temperature profile files (for example, tp\_100.xy).
- 7. Display contours of wall temperature for all six cases using the **wall-temperature** definition you created earlier.

**Results**  $\rightarrow$  Graphics  $\rightarrow$  Contours  $\rightarrow$  wall-temperature  $\xrightarrow{\Box}$  Display

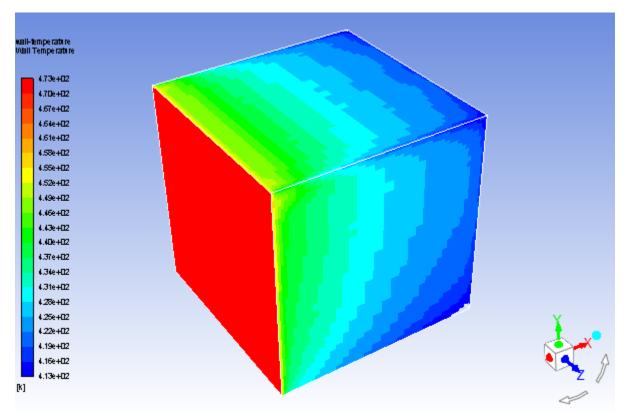
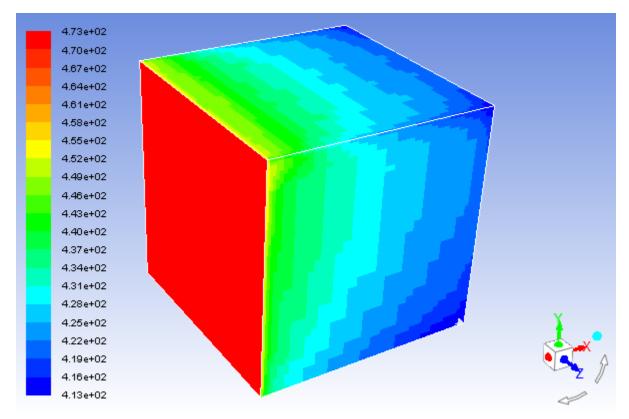


Figure 7.10: Contours of Wall Temperature: 1 Face per Surface Cluster





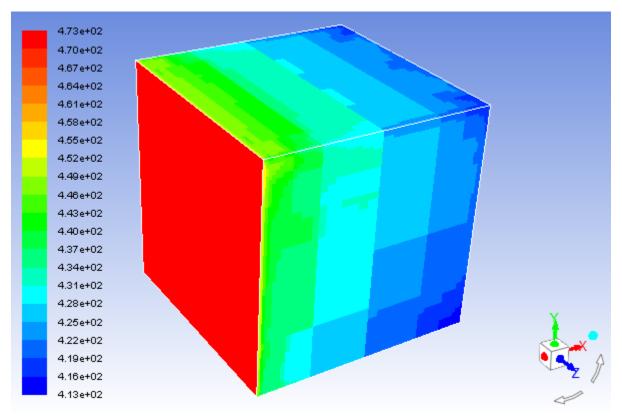
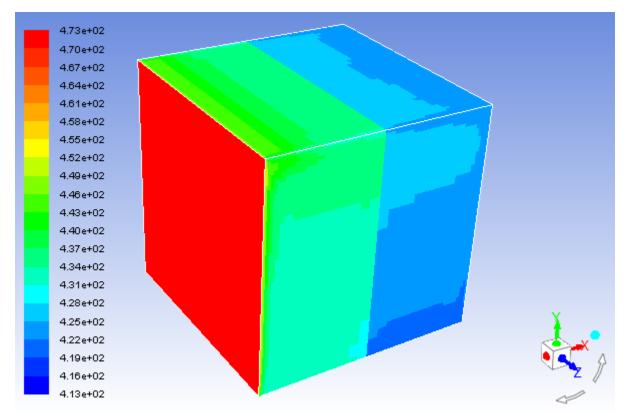


Figure 7.12: Contours of Wall Temperature: 100 Faces per Surface Cluster





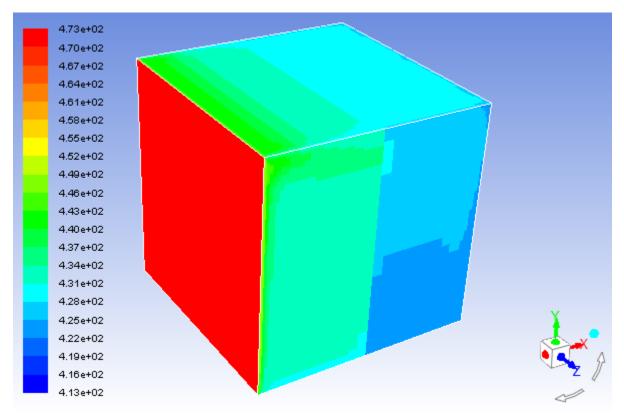
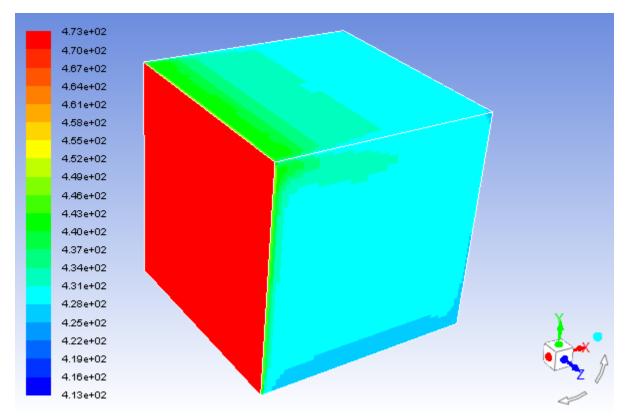


Figure 7.14: Contours of Wall Temperature: 800 Faces per Surface Cluster





8. Display contours of surface cluster ID for 1600 faces per surface cluster (Figure 7.16: Contours of **Surface Cluster ID**—1600 Faces per Surface Cluster (FPSC) (p. 354)).

Contours	
Options Filled	Contours of Radiation
Node Values	Surface Cluster ID 🗸
<ul> <li>Global Range</li> <li>Auto Range</li> </ul>	Min Max
Clip to Range	05
Draw Profiles Interpretation of the provided management of the pr	Surfaces Filter Text
ļ	default-interior w-high-x
Coloring	w-high-y
Banded Smooth	w-high-z w-low-x
Levels Setup	w-low-x w-low-y
20 💠 1 🚔	w-low-z
	zz_center_z zz_x_side
	New Surface 🔻
	Display Compute Close Help

Postprocessing  $\rightarrow$  Graphics  $\rightarrow$  Contours  $\rightarrow$  Edit...

- a. Ensure that the **Filled** option is enabled in the **Options** group box.
- b. Ensure that the Node Values option is disabled.
- c. Enable the Auto Range option.
- d. Select Radiation... and Surface Cluster ID from the Contours of drop-down lists.
- e. Ensure that all surfaces except **default-interior** and **zz\_x\_side** are selected in the **Surfaces** selection list.
- f. Click **Display** and rotate the view as shown in Figure 7.16: Contours of **Surface Cluster ID**—1600 Faces per Surface Cluster (FPSC) (p. 354).
- g. Close the **Contours** dialog box.

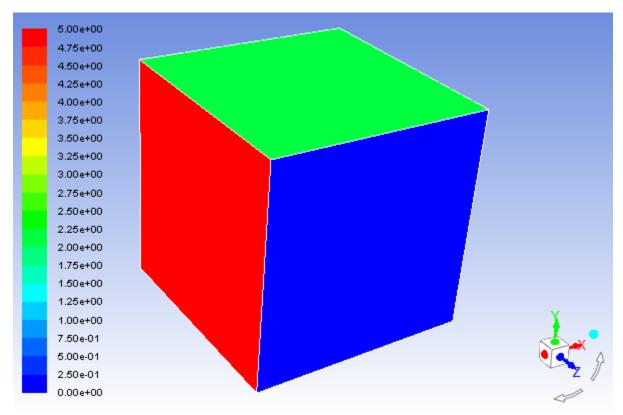


Figure 7.16: Contours of Surface Cluster ID—1600 Faces per Surface Cluster (FPSC)

9. Read rad\_400.cas.gz and rad\_400.dat.gz and, in a similar manner to the previous step, display contours of surface cluster ID (Figure 7.17: Contours of **Surface Cluster ID**—400 FPSC (p. 355)).

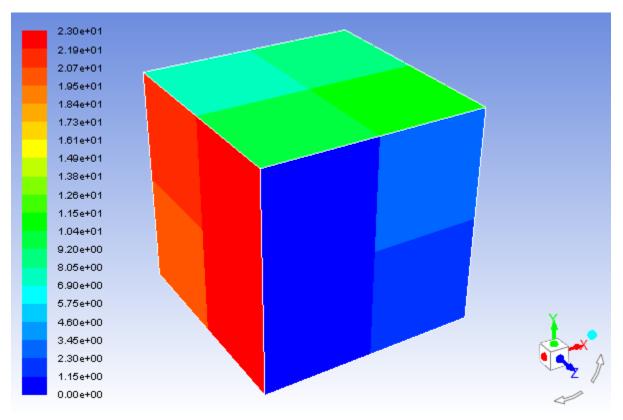


Figure 7.17: Contours of Surface Cluster ID—400 FPSC

*Figure 7.17: Contours of* **Surface Cluster ID**—400 FPSC (p. 355) shows contours of **Surface Cluster ID** for 400 FPSC. This case shows better clustering compared to all of the other cases.

10. Create a plot that compares the temperature profile plots for 1, 10, 100, 400, 800, and 1600 FPSC.

Postprocessing  $\rightarrow$  Plots  $\rightarrow$  File...

- a. Click the Add... button to open the Select File dialog box.
  - i. Select the file  $tp_1$ .xy that you created in step 11 of Postprocessing (p. 333).
  - ii. Click **OK** to close the **Select File** dialog box.
- b. Change the legend entry for the data series.

E File XY Plot		<b>X</b>
Plot Title Wall Temperature	Legend Title Faces/Clusters	Add
Files	Legend Entries	Delete Change Legend Entry
D:/Fluent Tutorials 180/Tut7/radiation_natural_convectio		
Plot Axes (	Curves Close Help	h.

- i. Enter Faces/Cluster in the Legend Title text box.
- ii. Enter 1 in the text box to the left of the **Change Legend Entry** button.
- iii. Click Change Legend Entry.

ANSYS Fluent will update the Legend Entry text for the file tp\_1.xy.

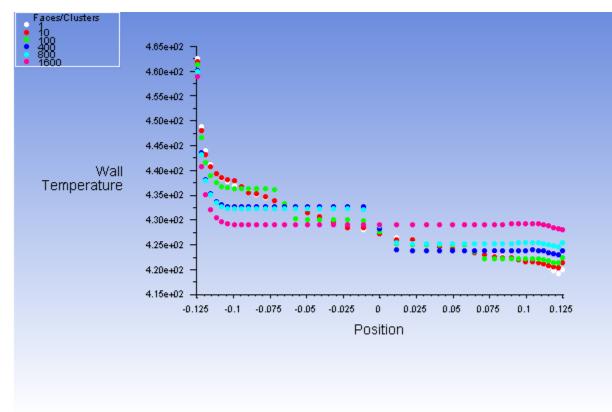
- c. Load the files tp\_10.xy, tp\_100.xy, tp\_400.xy, tp\_800.xy, and tp\_1600.xy and change their legend entries accordingly, in a manner similar to the previous two steps (a and b).
- d. Click the Axes... button to open the Axes dialog box.

💶 Axes - File XY Plot		×
Axis X Y Label	Number Format Type float Precision 3 Type Type Precision	Major Rules Color foreground Weight 1
Options Log Auto Range Major Rules Minor Rules	Range Minimum 0 Maximum 0	Minor Rules Color dark gray v Weight
	Apply Close Help	

- i. Ensure **X** is selected from the **Axis** list.
- ii. Enter 3 for Precision in the Number Format group box and click Apply.
- iii. Select Y from the Axis list.
- iv. Enter 2 for Precision and click Apply.

- v. Close the **Axes** dialog box.
- e. Click **Plot** (Figure 7.18: A Comparison of Temperature Profiles along the Outer Surface of the Box (p. 357)) and close the **File XY Plot** dialog box.





## 7.4.11. S2S Definition, Solution, and Postprocessing with Partial Enclosure

As mentioned previously, when the S2S model is used, you also have the option to define a "partial enclosure"; that is, you can disable the view factor calculation for walls with negligible emission/absorption, or walls that have uniform temperature. Even though the view factor will not be computed for these walls, they will still emit radiation at a fixed temperature called the "partial enclosure temperature". The main advantage of this is to speed up the view factor and the radiosity calculation.

In the steps that follow, you will specify the radiating wall (**w-low-x**) as a boundary zone that is not participating in the S2S radiation model. Consequently, you will specify the partial enclosure temperature for the wall. Note that the partial enclosure option may not yield accurate results in cases that have multiple wall boundaries that are not participating in S2S radiation and that each have different temperatures. This is because a single partial enclosure temperature is applied to all of the non-participating walls.

1. Read the case file saved previously for the S2S model (rad\_b\_1.cas.gz).

 $\blacksquare$  File  $\rightarrow$  Read  $\rightarrow$  Case...

2. Set the partial enclosure parameters for the S2S model.

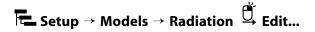
**The Setup**  $\rightarrow$  Boundary Conditions  $\rightarrow$  w-low-x  $\stackrel{\bigcirc}{\rightarrow}$  Edit...

🖸 Wall						<b>—</b> ×
Zone Name						
w-low-x						
Adjacent Cell Zone						
fluid						
Momentum Thermal Radiation	Species	DPM	Multiphase	UDS	Wall Film	Potential
S2S Parameters Faces Per Surface Cluster 1	1					
OK Cancel Help						

- a. Click the **Radiation** tab.
- b. Disable the Participates in View Factor Calculation option in the S2S Parameters group box.
- c. Click **OK** to close the **Wall** dialog box.

Click **OK** to close the dialog box informing you that you must recompute viewfactors.

3. Compute the view factors for the S2S model.



- a. Click the Settings... button to open the View Factors and Clustering dialog box.
- b. Click the Select... button to open the Participating Boundary Zones dialog box.

C.		
Participating Boundary Zones		
Maximum Distance (m) from Critical Zone		
To All Other Zones 0 Compute		
To Participating Zones 0 Apply		
Participating Boundary Zones Filter Text	To Ty Tx Non-Participating Boundary Zones Filter Te	xt 🔽 🔽 🗐
w-high-x	w-low-x	
w-high-y w-high-z		
w-low-y		
w-low-z		
Display Zones		
Non-Participating Boundary Zones Temperature (k) 473.15		
	OK Cancel Help	
	ok cancer Hep	

- i. Enter 473.15 K for Non-Participating Boundary Zones Temperature.
- ii. Click OK to close the Participating Boundary Zones dialog box.

Click **OK** to close the dialog box informing you that you must recompute viewfactors.

- c. Click OK to close the View Factors and Clustering dialog box.
- d. Click the **Compute/Write/Read...** button to open the **Select File** dialog box and to compute the view factors.

The view factor file will store the view factors for the radiating surfaces only. This may help you control the size of the view factor file as well as the memory required to store view factors in ANSYS Fluent. Furthermore, the time required to compute the view factors will be reduced, as only the view factors for radiating surfaces will be calculated.

- i. Enter rad\_partial.s2s.gz for S2S File.
- ii. Click **OK** in the **Select File** dialog box.
- e. Click OK to close the Radiation Model dialog box.
- 4. Right-click Solution Initialization and select Initialize.

**E**Solution  $\rightarrow$  Solution Initialization  $\stackrel{\square}{\hookrightarrow}$  Initialize

5. Start the calculation by requesting 400 iterations.

# **E** Solution $\rightarrow$ Run Calculation $\stackrel{\text{D}}{\rightarrow}$ Calculate

The solution will converge in approximately 300 iterations.

6. Save the case and data files (rad\_partial.cas.gz and rad\_partial.dat.gz).

File  $\rightarrow$  Write  $\rightarrow$  Case & Data...

7. Compute the radiation heat transfer rate.

Postprocessing $\rightarrow$ F	Reports → Fluxes
--------------------------------	------------------

E Flux Reports		
Options Mass Flow Rate	Boundaries Filter Text	Results
<ul> <li>Total Heat Transfer Rate</li> <li>Radiation Heat Transfer Rate</li> </ul>	default-interior w-high-x w-high-y w-high-z w-low-x w-low-y w-low-y	-11.20159577090498 -7.263738590544749 -10.93334593375261 -0 -12.68905165148573 -10.93108167311094
	•	• • •
Save Output Parameter		Net Results (w)
		-53.01881
	Compute Write Close Help	

- a. Ensure that Radiation Heat Transfer Rate is selected from the Options list.
- b. Select all boundary zones except **default-interior** from the **Boundaries** selection list.
- c. Click **Compute** and close the **Flux Reports** dialog box.

The **Flux Reports** dialog box does not report any heat transfer rate for the radiating wall (**w-low-x**), because you specified that it not participate in the view factor calculation. The remaining walls report similar rates to those obtained in step 9 of Postprocessing (p. 333), indicating that in this case the use of a partial enclosure saved computation time without significantly affecting the results.

8. Compare the temperature profile for the side wall to the profile saved in  $tp_1.xy$ .

Postprocessing  $\rightarrow$  Plots  $\rightarrow$  XY Plot  $\rightarrow$  Edit...

- a. Display the temperature profile for the side wall, **zz\_x\_side**, and write it to a file named tp\_partial.xy, in a manner similar to the instructions shown in step 13 of Postprocessing (p. 333).
- b. Click Load File... to open the Select File dialog box.
  - i. Select **tp\_1.xy**.
  - ii. Click **OK** to close the **Select File** dialog box.
- c. Click Plot.
- d. Close the **Solution XY Plot** dialog box.

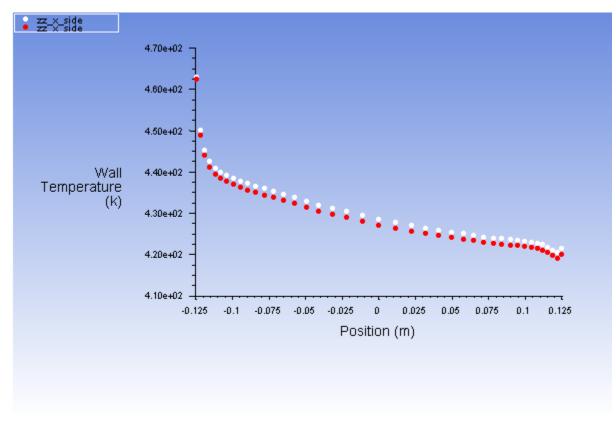


Figure 7.19: Wall Temperature Profile Comparison

*Figure 7.19: Wall Temperature Profile Comparison (p. 361) further confirms that the use of a partial enclosure did not significantly affect the results.* 

## 7.5. Summary

In this tutorial you studied combined natural convection and radiation in a three-dimensional square box and compared how varying the settings of the surface-to-surface (S2S) radiation model affected the results. The S2S radiation model is appropriate for modeling the enclosure radiative transfer without participating media, whereas the methods for participating radiation may not always be efficient.

For more information about the surface-to-surface (S2S) radiation model, see the Fluent User's Guide.

## 7.6. Further Improvements

This tutorial guides you through the steps to reach an initial solution. You may be able to obtain a more accurate solution by using an appropriate higher-order discretization scheme and by adapting the mesh. Mesh adaption can also ensure that the solution is independent of the mesh. These steps are demonstrated in Introduction to Using ANSYS Fluent: Fluid Flow and Heat Transfer in a Mixing Elbow (p. 121).

# **Chapter 8: Modeling Flow Through Porous Media**

This tutorial is divided into the following sections:

- 8.1. Introduction
- 8.2. Prerequisites
- 8.3. Problem Description
- 8.4. Setup and Solution
- 8.5. Summary
- 8.6. Further Improvements

## 8.1. Introduction

Many industrial applications such as filters, catalyst beds, and packing, involve modeling the flow through porous media. This tutorial illustrates how to set up and solve a problem involving gas flow through porous media.

The industrial problem solved here involves gas flow through a catalytic converter. Catalytic converters are commonly used to purify emissions from gasoline and diesel engines by converting environmentally hazardous exhaust emissions to acceptable substances. Examples of such emissions include carbon monoxide (CO), nitrogen oxides (NOx), and unburned hydrocarbon fuels. These exhaust gas emissions are forced through a substrate, which is a ceramic structure coated with a metal catalyst such as platinum or palladium.

The nature of the exhaust gas flow is a very important factor in determining the performance of the catalytic converter. Of particular importance is the pressure gradient and velocity distribution through the substrate. Hence, CFD analysis is useful for designing efficient catalytic converters. By modeling the exhaust gas flow, the pressure drop and the uniformity of flow through the substrate can be determined. In this tutorial, ANSYS Fluent is used to model the flow of nitrogen gas through a catalytic converter geometry, so that the flow field structure may be analyzed.

This tutorial demonstrates how to do the following:

- Set up a porous zone for the substrate with appropriate resistances.
- Calculate a solution for gas flow through the catalytic converter using the pressure-based solver.
- Plot pressure and velocity distribution on specified planes of the geometry.
- Determine the pressure drop through the substrate and the degree of non-uniformity of flow through cross sections of the geometry using X-Y plots and numerical reports.

## 8.2. Prerequisites

This tutorial is written with the assumption that you have completed one or more of the introductory tutorials found in this manual:

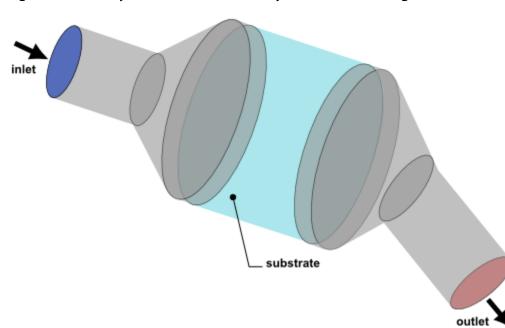
• Introduction to Using ANSYS Fluent in ANSYS Workbench: Fluid Flow and Heat Transfer in a Mixing Elbow (p. 1)

- Parametric Analysis in ANSYS Workbench Using ANSYS Fluent (p. 73)
- Introduction to Using ANSYS Fluent: Fluid Flow and Heat Transfer in a Mixing Elbow (p. 121)

and that you are familiar with the ANSYS Fluent tree and ribbon structure. Some steps in the setup and solution procedure will not be shown explicitly.

## 8.3. Problem Description

The catalytic converter modeled here is shown in Figure 8.1: Catalytic Converter Geometry for Flow Modeling (p. 364). The nitrogen flows through the inlet with a uniform velocity of 22.6 m/s, passes through a ceramic monolith substrate with square-shaped channels, and then exits through the outlet.



#### Figure 8.1: Catalytic Converter Geometry for Flow Modeling

While the flow in the inlet and outlet sections is turbulent, the flow through the substrate is laminar and is characterized by inertial and viscous loss coefficients along the inlet axis. The substrate is impermeable in other directions. This characteristic is modeled using loss coefficients that are three orders of magnitude higher than in the main flow direction.

## 8.4. Setup and Solution

The following sections describe the setup and solution steps for this tutorial:

8.4.1. Preparation
8.4.2. Mesh
8.4.3. General Settings
8.4.4. Models
8.4.5. Materials
8.4.6. Cell Zone Conditions
8.4.7. Boundary Conditions
8.4.8. Solution
8.4.9. Postprocessing

## 8.4.1. Preparation

To prepare for running this tutorial:

- 1. Set up a working folder on the computer you will be using.
- 2. Go to the ANSYS Customer Portal, https://support.ansys.com/training.

#### Note

If you do not have a login, you can request one by clicking **Customer Registration** on the log in page.

- 3. Enter the name of this tutorial into the search bar.
- 4. Narrow the results by using the filter on the left side of the page.
  - a. Click ANSYS Fluent under Product.
  - b. Click 18.0 under Version.
- 5. Select this tutorial from the list.
- 6. Click the **porous\_R180.zip** link to download the input files.
- 7. Unzip porous\_R180.zip to your working folder.

The mesh file catalytic\_converter.msh can be found in the porous directory created after unzipping the file.

8. Use Fluent Launcher to start the **3D** version of ANSYS Fluent, with the **Double Precision** and **Display Mesh After Reading** options enabled.

For more information about starting ANSYS Fluent using the Fluent Launcher, see the Fluent Getting Started Guide, available in the help viewer or on the ANSYS Customer Portal (http://support.an-sys.com/documentation).

### 8.4.2. Mesh

1. Read the mesh file (catalytic\_converter.msh).

File → Read → Mesh...

2. Check the mesh.

Setting Up Domain  $\rightarrow$  Mesh  $\rightarrow$  Check

ANSYS Fluent will perform various checks on the mesh and report the progress in the console. Make sure that the reported minimum volume is a positive number.

3. Scale the mesh.

**Setting Up Domain**  $\rightarrow$  Mesh  $\rightarrow$  Scale...

Scale Mesh		<b>X</b>				
Domain Extents		Scaling				
Xmin (mm) -1.22461e-15	Xmax (mm) 276.7358	Convert Units				
Ymin (mm) -58.32051	Ymax (mm) 50	Specify Scaling Factors				
Zmin (mm) -50	Zmax (mm) 50	Mesh Was Created In				
View Length Unit In mm 🔹		Scaling Factors           X         0.001           Y         0.001           Z         0.001           Scale         Unscale				
Close Help						

- a. Select mm from the Mesh Was Created In drop-down list.
- b. Click Scale.
- c. Select mm from the View Length Unit In drop-down list.

All dimensions will now be shown in millimeters.

- d. Close the Scale Mesh dialog box.
- 4. Check the mesh.

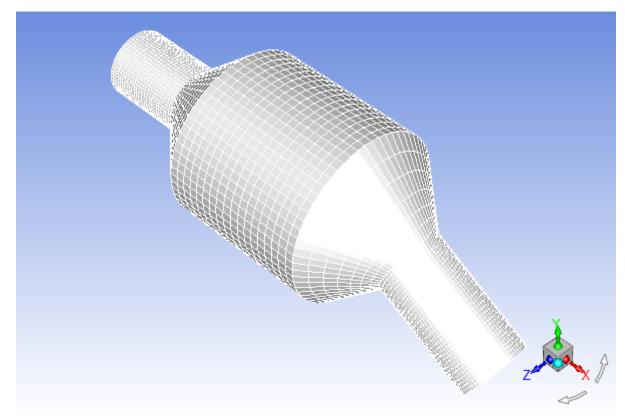
Setting Up Domain  $\rightarrow$  Mesh  $\rightarrow$  Check

#### Note

It is a good idea to check the mesh after you manipulate it (that is, scale, convert to polyhedra, merge, separate, fuse, add zones, or smooth and swap.) This will ensure that the quality of the mesh has not been compromised.

5. Examine the mesh.

Rotate the view and zoom in to get the display shown in Figure 8.2: Mesh for the Catalytic Converter Geometry (p. 367). The hex mesh on the geometry contains a total of 34,580 cells.



### Figure 8.2: Mesh for the Catalytic Converter Geometry

## 8.4.3. General Settings

## Setting Up Physics → Solver

		Solver	
Time Steady	Type Pressure-Based	Velocity Formulation	Operating Conditions
<ul> <li>Transient</li> </ul>	<ul> <li>Density-Based</li> </ul>	<ul> <li>Relative</li> </ul>	🥏 Reference Values

1. Retain the default solver settings.

## 8.4.4. Models

1. Select the standard k- $\varepsilon$  turbulence model.

**Setting Up Physics**  $\rightarrow$  Models  $\rightarrow$  Viscous...

Viscous Model	
Model Inviscid Laminar Spalart-Allmaras (1 eqn) K-epsilon (2 eqn) K-omega (2 eqn) Transition k-kl-omega (3 eqn) Transition SST (4 eqn) Reynolds Stress (7 eqn) Scale-Adaptive Simulation (SAS) Detached Eddy Simulation (DES) Large Eddy Simulation (LES)	Model Constants Cmu 0.09 C1-Epsilon 1.44 C2-Epsilon 1.92 TKE Prandtl Number 1 TDR Prandtl Number 1.3
<ul> <li>k-epsilon Model</li> <li>Standard</li> <li>RNG</li> <li>Realizable</li> <li>Near-Wall Treatment</li> <li>Standard Wall Functions</li> <li>Scalable Wall Functions</li> <li>Non-Equilibrium Wall Functions</li> <li>Enhanced Wall Treatment</li> <li>Menter-Lechner</li> <li>User-Defined Wall Functions</li> </ul>	User-Defined Functions Turbulent Viscosity none Prandtl Numbers TKE Prandtl Number none TDR Prandtl Number none TDR Prandtl Number
Options Curvature Correction Production Kato-Launder Production Limiter OK	ancel Help

a. Select **k-epsilon (2eqn)** in the **Model** list.

The original **Viscous Model** dialog box will now expand.

b. Retain the default settings for **k-epsilon Model** and **Near-Wall Treatment** and click **OK** to close the **Viscous Model** dialog box.

### 8.4.5. Materials

1. Add nitrogen to the list of fluid materials by copying it from the **Fluent Database** of materials.

**Setting Up Physics**  $\rightarrow$  Materials  $\rightarrow$  Create/Edit...

Create/Edit Materials				<b>—</b> ———————————————————————————————————	
Name		Material Type	Material Type		
air		fluid	-	Name	
Chemical Formula		Fluent Fluid Materials		Chemical Formula	
		air	•	Fluent Database	
		Mixture		User-Defined Database	
		none	Ψ	User-Defined Database	
Properties					
Density (kg/m3)	constant	- Edit			
	1.225				
Viscosity (kg/m-s)	constant	▼ Edit			
	1.7894e-05				
		Change/Create Delete Close He	elp		

a. Click the Fluent Database... button to open the Fluent Database Materials dialog box.

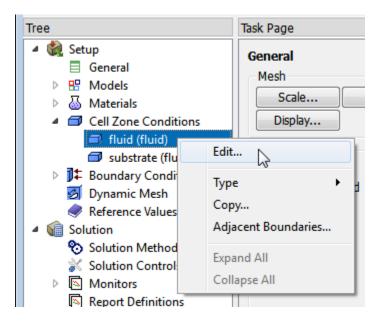
Fluent Database Materials	
Fluent Fluid Materials [1/563]	Material Type fluid
nitro-methylene (h2cno2)  nitro-silane (h3sin)	Order Materials by Name
nitrogen (n2)	Chemical Formula
nitrogen-dihydride (nh2)	
nitrogen-dioxide (no2)	
nitrogen-dioxide- (no2-)	
Copy Materials from Case Delete Properties	
Density (kg/m3) constant	▼ View ▲
1.138	E
Cp (Specific Heat) (j/kg-k) piecewise-polynomial	▼ View
Thermal Conductivity (w/m-k)	View
0.0242	
Viscosity (kg/m-s) constant	View
1.663e-05	
	•
New Edit Save Copy Clos	se Help

- i. Select nitrogen (n2) in the Fluent Fluid Materials selection list.
- ii. Click **Copy** to copy the information for nitrogen to your list of fluid materials.

- iii. Close the Fluent Database Materials dialog box.
- b. Click Change/Create and close the Create/Edit Materials dialog box.

### 8.4.6. Cell Zone Conditions

**The Setup**  $\rightarrow$  Cell Zone Conditions  $\rightarrow$  fluid  $\stackrel{0}{\hookrightarrow}$  Edit...



1. Set the cell zone conditions for the fluid (**fluid**).

E Fluid							<b>X</b>	
Zone Name								
fuid								
	Edit							
Frame Motion J 3D Fan Zone Source Terms Hesh Motion Frame Motion Frame Terms Frame Motion Frame Terms Frame Motion Frame Terms Frame Motion Frame M								
Porous Zone	rixeu values							
Reference Frame Mesh Motion	Porous Zone	3D Fan Zone	Embedded LES	Reaction	Source Terms	Fixed Values	Multiphase	
Rotation-Axis Origin		Rotation-Axis D	Direction					
X (mm) 0 constant	•	x 0	constant 💌					
Y (mm) 0 constant	*	Y 0	constant	•				
Z (mm) 0 constant	•	Z 1	constant	•				
		ОК	Cancel Help					
							.::	

- a. Select nitrogen from the Material Name drop-down list.
- b. Click **OK** to close the **Fluid** dialog box.

2. Set the cell zone conditions for the substrate (substrate).

Eluid IIII						
ione Name substrate						-
Naterial Name (nitrogen 💌 ) E	dit					
Frame Motion 📃 3D Fan Zone 📃 Source	e Terms					
Mesh Motion 🗹 Laminar Zone 🗐 Fixed	Values					
Porous Zone						
Reference Frame Mesh Motion Poro	us Zone 3D Fan Zone	Embedded LES	Reaction	Source Terms	Fixed Values	Multiphase
Conical Conical Relative Velocity Resistance Formulation Viscous Resistance (Inverse Absolute Perr Direction-1 (1/m2) 3.846e+07	neability)	*				1
Direction-2 (1/m2) 3.846e+10	constant	•				I
Direction-3 (1/m2) 3.846e+10	constant	-				I
Inertial Resistance Alternative Formulation Direction-1 (1/m) 20.414 Direction-2 (1/m) 20.414 Direction-3 (1/m) 20.414	constant constant constant					

- a. Select nitrogen from the Material Name drop-down list.
- b. Enable **Porous Zone** to activate the porous zone model.
- c. Enable Laminar Zone to solve the flow in the porous zone without turbulence.
- d. Click the **Porous Zone** tab.
  - i. Make sure that the principal direction vectors are set as shown in Table 8.1: Values for the Principle Direction Vectors (p. 371).

ANSYS Fluent automatically calculates the third (Z direction) vector based on your inputs for the first two vectors. The direction vectors determine which axis the viscous and inertial resistance coefficients act upon.

Axis	Direction-1 Vector	Direction-2 Vector
Х	1	0
Y	0	1
Z	0	0

ii. For the viscous and inertial resistance directions, enter the values in Table 8.2: Values for the Viscous and Inertial Resistance (p. 372) Viscous Resistance and Inertial Resistance.

*Direction-2* and *Direction-3* are set to arbitrary large numbers. These values are several orders of magnitude greater than that of the Direction-1 flow and will make any radial flow insignificant.

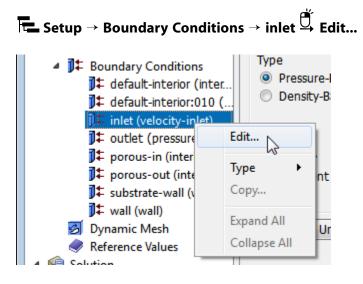
Scroll down to access the fields that are not initially visible.

Table 8.2: Values for the Viscous and Inertial Resistance

Direction	Viscous Resistance (1/m2)	Inertial Resistance (1/m)		
Direction-1	3.846e+07	20.414		
Direction-2	3.846e+10	20414		
Direction-3	3.846e+10	20414		

e. Click **OK** to close the **Fluid** dialog box.

## 8.4.7. Boundary Conditions



1. Set the velocity and turbulence boundary conditions at the inlet (inlet).

Velocity Inlet								<b>—</b> ×
Zone Name						_		
inlet								
Momentum	Thermal	Radiation	Species	DPM	Multipl	nase	Potential	UDS
Velocit	y Specificatio	on Method Ma	agnitude, N	ormal to B	Boundar	/		•
	Refere	nce Frame Ab	osolute					•
Velocity Magnitude (m/s) 22.6 constant								•
Supersonic/Initial Gauge Pressure (pascal) 0 constant								
	- Turbulence							
	Specification	n Method Int	ensity and H	Hydraulic [	Diamete	r		•
			Turbulent	Intensity (	(%) 10			P
			Hydraulic Di	ameter (n	nm) 42			P
		O	Cancel	Help				$\searrow$

- a. Enter 22.6 m/s for **Velocity Magnitude**.
- b. In the **Turbulence** group box, select **Intensity and Hydraulic Diameter** from the **Specification Method** drop-down list.
- c. Enter 10% for the Turbulent Intensity.
- d. Enter 42 mm for the Hydraulic Diameter.
- e. Click **OK** to close the **Velocity Inlet** dialog box.
- 2. Set the boundary conditions at the outlet (outlet).



Pressure Out	let						<b>—</b> ———————————————————————————————————	
Zone Name								
outlet								
Momentum	Thermal	Radiation	Species	DPM	Multiphase	Potential	UDS	
Backflow Reference Frame Absolute								
Gauge Pressure (pascal) 0						constant	<b>_</b>	
Backflow Direction Specification Method Normal to Boundary								
Backflow Pressure Specification Total Pressure								
Radial Equilibrium Pressure Distribution								
Average Pr	essure Specif	ication						
Target Mass	s Flow Rate							
	- Turbulen	се						
	Specificat	ion Method 🛛	Intensity and	d Hydraul	ic Diameter		<b>_</b>	
		Backflow Turbulent Intensity (%)					P	
	Backflow Hydraulic Diameter (mm)						P	
OK Cancel Help								

- a. Retain the default setting of **0** for **Gauge Pressure**.
- b. In the **Turbulence** group box, select **Intensity and Hydraulic Diameter** from the **Specification Method** drop-down list.
- c. Retain the default value of 5% for the **Backflow Turbulent Intensity**.
- d. Enter 42 mm for the **Backflow Hydraulic Diameter**.
- e. Click **OK** to close the **Pressure Outlet** dialog box.
- 3. Retain the default boundary conditions for the walls (substrate-wall and wall).

## 8.4.8. Solution

1. Set the solution parameters.



Solution Methods
Pressure-Velocity Coupling
Scheme
Coupled
Spatial Discretization
Gradient
Least Squares Cell Based 🔹
Pressure
Second Order
Momentum
Second Order Upwind
Turbulent Kinetic Energy
First Order Upwind
Turbulent Dissipation Rate
First Order Upwind
Transient Formulation
Non-Iterative Time Advancement
Frozen Flux Formulation
Pseudo Transient
Warped-Face Gradient Correction
High Order Term Relaxation Options
Default

- a. Select **Coupled** from the **Scheme** drop-down list.
- b. Retain the default selection of **Least Squares Cell Based** from the **Gradient** drop-down list in the **Spatial Discretization** group box.
- c. Retain the default selection of **Second Order Upwind** from the **Momentum** drop-down list.
- d. Enable Pseudo Transient.
- 2. Enable the plotting of the mass flow rate at the outlet.

**Solving**  $\rightarrow$  Reports  $\rightarrow$  Definitions  $\rightarrow$  New  $\rightarrow$  Surface Report  $\rightarrow$  Mass Flow Rate

Surface Report Definition	
Name	Report Type
surf-mon-1	Mass Flow Rate
Options	Custom Vectors Vectors of
Average Over	Custom Vectors Field Variable
Report Files [0/0]	Pressure v
	Static Pressure
	Surfaces Filter Text
Report Plots [0/0]	default-interior default-interior:010 inlet
	outlet
	porous-in
	porous-out
	substrate-wall wall
Create	
☑ Report File	
🔽 Report Plot	
Frequency 1	
V Print to Console	Highlight Surfaces
Create Output Parameter	New Surface 💌
OK	Compute Cancel Help

- a. Enter **surf-mon-1** for the **Name** of the surface report definition.
- b. In the Create group box, enable Report File, Report Plot and Print to Console.
- c. Select outlet in the Surfaces selection list.
- d. Click **OK** to save the surface report definition settings and close the **Surface Report Definition** dialog box.
- 3. Initialize the solution from the inlet.

<b>Solving</b> $\rightarrow$ Initialization									
Initialization									
Method Mybrid		Patch	t = 0						
<ul> <li>Nybrid</li> <li>Standard</li> </ul>	More Settings	Reset Statistics							
Scandard	Options	Reset DPM	Initialize						

a. Select Standard under Method.

#### Warning

**Standard** is the recommended initialization method for porous media simulations. The default **Hybrid** method does not account for the porous media properties, and depending on boundary conditions, may produce an unrealistic initial velocity field. For porous media simulations, the **Hybrid** method should only be used when the **Maintain Constant Velocity Magnitude** option is enabled in the **Hybrid Initialization** dialog box.

b. Click Options... to open the Solution Initialization task page, which provides access to further settings.

Solution Initialization	
Initialization Methods	
O Hybrid Initialization	
Standard Initialization	
Compute from	
inlet 🔹	
Reference Frame	
Relative to Cell Zone	
Absolute	
Initial Values	_
Gauge Pressure (pascal)	
0	
X Velocity (m/s)	
22.6	
Y Velocity (m/s)	
-1.340561e-15	
Z Velocity (m/s)	
-1.324296e-32	
Turbulent Kinetic Energy (m2/s2)	
7.6614	
Turbulent Dissipation Rate (m2/s3)	
1185.214	
Initialize Reset Patch	
Reset DPM Sources Reset Statistics	

- i. Select **inlet** from the **Compute from** drop-down list in the **Solution Initialization** task page.
- ii. Retain the default settings for standard initialization method.

- iii. Click Initialize.
- 4. Save the case file (catalytic\_converter.cas.gz).

**File**  $\rightarrow$  Write  $\rightarrow$  Case...

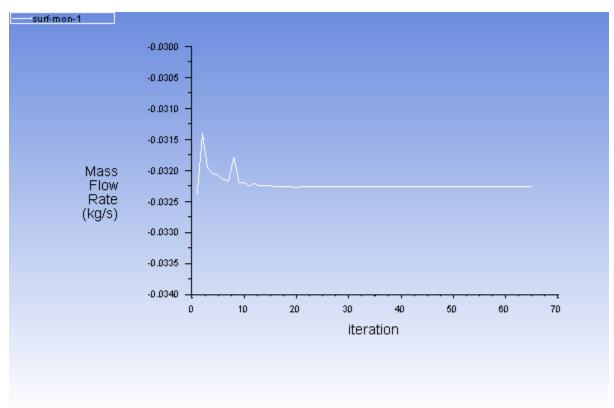
5. Run the calculation by requesting 100 iterations.

Solving → Run Calculation

- a. Enter 100 for **No. of Iterations**.
- b. Click **Calculate** to begin the iterations.

The solution will converge in approximately 65 iterations. The mass flow rate graph flattens out, as seen in Figure 8.3: Mass Flow Rate History (p. 378).

Figure 8.3: Mass Flow Rate History



6. Save the case and data files (catalytic\_converter.cas.gz and catalytic\_converter.dat.gz).



#### Note

If you choose a filename that already exists in the current folder, ANSYS Fluent will prompt you for confirmation to overwrite the file.

### 8.4.9. Postprocessing

1. Create a surface passing through the centerline for postprocessing purposes.

### Postprocessing $\rightarrow$ Surface $\rightarrow$ Create $\rightarrow$ Iso-Surface...

Iso-Surface		
Surface of Constant Mesh	-	From Surface Filter Text
Y-Coordinate	•	
Min (mm)	Max (mm) 50	default-interior:010 inlet ≡ outlet
Iso-Values (mm) 0		porous-in porous-out
< □	<b>&gt;</b>	From Zones Fiter Text
New Surface Name		fluid substrate
	Create	Manage Close Help

- a. Select Mesh... and Y-Coordinate from the Surface of Constant drop-down lists.
- b. Click **Compute** to calculate the **Min** and **Max** values.
- c. Retain the default value of **0** for **Iso-Values**.
- d. Enter y=0 for **New Surface Name**.
- e. Click Create.

#### Note

To interactively place the surface on your mesh, use the slider bar in the **Iso-Surface** dialog box.

2. Create cross-sectional surfaces at locations on either side of the substrate, as well as at its center.



Iso-Surface		
Surface of Constant Mesh		From Surface Filter Text
X-Coordinate		- default-interior
Min (mm)	Max (mm)	default-interior:010 inlet  ≡
-1.224606e-15 Iso-Values (mm)	276.7358	outlet
95		porous-out +
◄ □		From Zones Filter Text
New Surface Name		fluid
x=95		substrate
	Create Compu	te Manage Close Help

- a. Select Mesh... and X-Coordinate from the Surface of Constant drop-down lists.
- b. Click **Compute** to calculate the **Min** and **Max** values.
- c. Enter 95 for Iso-Values.
- d. Enter x=95 for the **New Surface Name**.
- e. Click Create.
- f. In a similar manner, create surfaces named x=130 and x=165 with **Iso-Values** of 130 and 165, respectively.
- g. Close the **Iso-Surface** dialog box after all the surfaces have been created.
- 3. Create a line surface for the centerline of the porous media.

Postprocessing  $\rightarrow$  Surface  $\rightarrow$  Create  $\rightarrow$  Line/Rake...

Line/Ra	ke Surface			Number of Points
Line T Reset		Type Line		
-End Poin	ts			
x0 (mm)	95		x1 (mm)	165
y0 (mm)	0		y1 (mm)	0
z0 (mm)	0		z1 (mm)	0
Select Points with Mouse				
New Surfa	ce Name			
porous-cl				
Create Manage Close Help				

- a. Enter the coordinates of the end points of the line in the **End Points** group box as shown.
- b. Enter porous-cl for the New Surface Name.
- c. Click **Create** to create the surface.
- d. Close the Line/Rake Surface dialog box.
- 4. Display the two wall zones (substrate-wall and wall).

Postprocessing  $\rightarrow$  Graphics  $\rightarrow$  Mesh...

💶 Mesh Display	y	×	
Options Nodes	Edge Type	Surfaces Filter Text	]
Edges	Feature	default-interior:010	
Faces	Outline	inlet	
Partitions		outlet	
Overset		porous-cl 🗉	
Shrink Factor	Eastura Angla	porous-in	
	Feature Angle	porous-out	
0	20	substrate-wall	
Outline	Interior	wall	
Adjacency		New Surface	-
	Di	splay Colors Close Help	
	D		111

a. Disable **Edges** and make sure **Faces** is enabled in the **Options** group box.

b. Deselect inlet and outlet in the Surfaces selection list, and make sure that only substrate-wall and wall are selected.

х

Close

Set

Ŧ

- c. Click **Display** and close the **Mesh Display** dialog box.
- 5. Set the lighting for the display.

Γ.

$\blacksquare Viewing \rightarrow Display \rightarrow Options$	•••
Display Options	
Rendering Line Width 1 Point Symbol (+) Animation Option All	Graphics Window Active Window 2 2 Se Color Scheme Workbench
<ul> <li>Double Buffering</li> <li>Outer Face Culling</li> <li>Hidden Line Removal</li> <li>Hidden Surface Removal</li> <li>Removal Method</li> </ul>	Lighting Attributes Lights On Lighting Gouraud Layout

	Lighting 🔒
Hidden Surface Removal	Gouraud 🔻
Removal Method Hardware Z-buffer	Layout
Display Timeout	Titles
	Axes
Timeout in seconds 60	Ruler
	🔲 Logo
	Color White 💌
	Colormap
	Colormap Alignment
Apply Info Lights	Close Help

- a. Disable **Double Buffering** in the **Rendering** group box.
- Make sure Lights On is enabled in the Lighting Attributes group box. b.
- Select **Gouraud** from the **Lighting** drop-down list. c.
- d. Click Apply and close the Display Options dialog box.
- 6. Set the transparency parameter for the wall zones (substrate-wall and wall).

 $\blacksquare$  Viewing  $\rightarrow$  Graphics  $\rightarrow$  Compose...

Scene Description		<b>×</b>
Names [2/2]	Geometry Attributes Type Group Display Transform Iso-Value Pathlines Time Step	Scene Composition Overlays Draw Frame Frame Options
A	pply Close Help	

- a. Select **substrate-wall** and **wall** in the **Names** selection list.
- b. Click the **Display...** button in the **Geometry Attributes** group box to open the **Display Properties** dialog box.

Display Properties	×	
Geometry Name		
Group		
Visibility	Colors	
Visible	Color	
Lighting	face-color 🔹	
Faces	⊿ ≥ 255	
Outer Faces		
Edges	Red	
Perimeter Edges	✓ ► <sup>255</sup>	
Feature Edges Lines	Green	
Nodes	255	
	Blue	
	✓ — 70	
	Transparency	
Apply Close Help		
(App)		

- i. Make sure that the **Red**, **Green**, and **Blue** sliders are set to the maximum position (that is, **255**).
- ii. Set the **Transparency** slider to **70**.
- iii. Click **Apply** and close the **Display Properties** dialog box.

- c. Click **Apply** and close the **Scene Description** dialog box.
- 7. Display velocity vectors on the **y=0** surface (Figure 8.4: Velocity Vectors on the y=0 Plane (p. 385)).

i ostprocessing Graphics vectors Luit	cessing $\rightarrow$ Graphics $\rightarrow$ Vectors $\rightarrow$ Edit
---------------------------------------	---

Vectors	×
Options <ul> <li>Options</li> <li>Global Range</li> <li>Auto Range</li> <li>Clip to Range</li> </ul>	Vectors of Velocity Color by
<ul> <li>Auto Scale</li> <li>Draw Mesh</li> </ul>	Velocity  Velocity Magnitude  Min Max
Style arrow	
Scale Skip 5 1 🚔	Surfaces Filter Text
Vector Options	substrate-wall wall
Custom Vectors	x=95 y-coordinate-9
	New Surface
	Display Compute Close Help

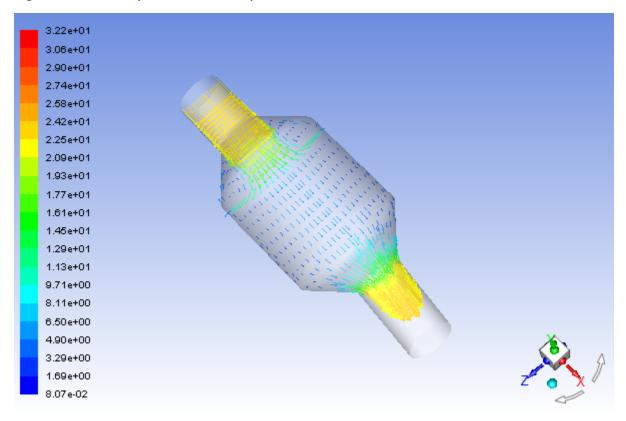
a. Enable **Draw Mesh** in the **Options** group box to open the **Mesh Display** dialog box.

💶 Mesh Display	/		×
Options Nodes	<ul> <li>Edge Type</li> <li>All</li> </ul>	Surfaces Filter Text	
Edges	Feature	default-interior:010	*
Faces	Outline	inlet	
Partitions		outlet	
Overset		porous-cl	E
Shrink Factor	Feature Angle	porous-in porous-out	
0	20	substrate-wall	
Outline	Interior	wall	
Adjacency		New Surface 🔻	
	D	isplay Colors Close Help	н. Н

i. Make sure that substrate-wall and wall are selected in the Surfaces selection list.

- ii. Click **Display** and close the **Mesh Display** dialog box.
- b. Enter 5 for Scale.
- c. Set Skip to 1.
- d. Select **y=0** in the **Surfaces** selection list.
- e. Click **Display** and close the **Vectors** dialog box.
- f. Rotate the view and adjust the magnification to get the display shown in Figure 8.4: Velocity Vectors on the y=0 Plane (p. 385).

Figure 8.4: Velocity Vectors on the y=0 Plane



The flow pattern shows that the flow enters the catalytic converter as a jet, with recirculation on either side of the jet. As it passes through the porous substrate, it decelerates and straightens out, and exhibits a more uniform velocity distribution. This allows the metal catalyst present in the substrate to be more effective.

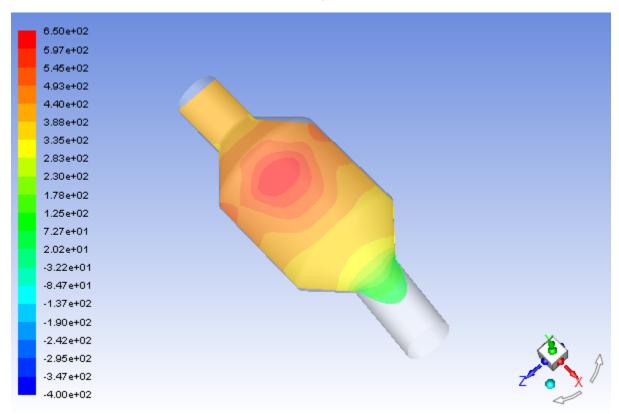
8. Display filled contours of static pressure on the **y=0** plane (Figure 8.5: Contours of Static Pressure on the y=0 plane (p. 387)).

Postprocessing  $\rightarrow$  Graphics  $\rightarrow$  Contours  $\rightarrow$  Edit...

Contours	
Options Filled	Contours of Pressure
<ul> <li>Node Values</li> <li>Global Range</li> </ul>	Static Pressure
Auto Range Clip to Range	Min Max 0 0
<ul><li>Draw Profiles</li><li>Draw Mesh</li></ul>	Surfaces Filter Text
Coloring Banded Smooth	porous-in porous-out substrate-wall wall x=95
Levels Setup 20 🜩 1 🜩	y-coordinate-9 y=0 New Surface
	Display Compute Close Help

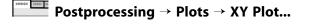
- a. Enable **Filled** in the **Options** group box.
- b. Enable Draw Mesh to open the Mesh Display dialog box.
  - i. Make sure that substrate-wall and wall are selected in the Surfaces selection list.
  - ii. Click **Display** and close the **Mesh Display** dialog box.
- c. Make sure that Pressure... and Static Pressure are selected from the Contours of drop-down lists.
- d. Select **y=0** in the **Surfaces** selection list.
- e. Click **Display** and close the **Contours** dialog box.

The pressure changes rapidly in the middle section, where the fluid velocity changes as it passes through the porous substrate. The pressure drop can be high, due to the inertial and viscous resistance of the porous media. Determining this pressure drop is one of the goals of the CFD analysis. In the next step, you will learn how to plot the pressure drop along the centerline of the substrate.



### Figure 8.5: Contours of Static Pressure on the y=0 plane

9. Plot the static pressure across the line surface **porous-cl** (Figure 8.6: Plot of Static Pressure on the porous-cl Line Surface (p. 388)).



Solution XY Plot			<b>—</b> ———————————————————————————————————		
Options          Options         Image: Node Values         Image: Position on X Axis         Image: Position on Y Axis         Image: Write to File         Image: Order Points	Plot D X 1 Y 0 Z 0 Load F Free D	default-interior:010			
Plot Axes Curves Close Help					

Release 18.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- a. Make sure that Pressure... and Static Pressure are selected from the Y Axis Function drop-down lists.
- b. Select **porous-cl** in the **Surfaces** selection list.
- c. Click **Plot** and close the **Solution XY Plot** dialog box.

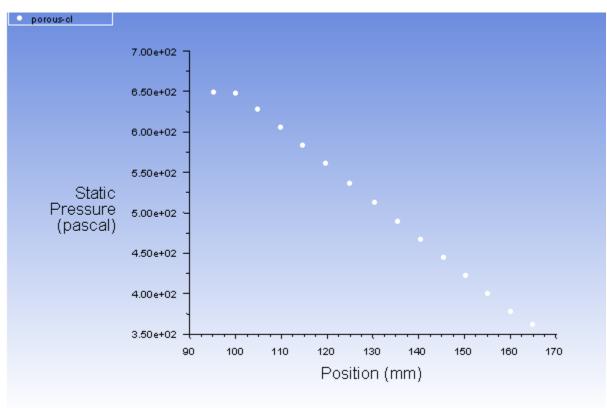


Figure 8.6: Plot of Static Pressure on the porous-cl Line Surface

As seen in Figure 8.6: Plot of Static Pressure on the porous-cl Line Surface (p. 388), the pressure drop across the porous substrate is approximately 300 Pa.

10. Display filled contours of the velocity in the X direction on the **x=95**, **x=130**, and **x=165** surfaces (Figure 8.7: Contours of the X Velocity on the x=95, x=130, and x=165 Surfaces (p. 390)).



Contours	
Options Filled	Contours of Velocity
Node Values Global Range	X Velocity 🗸
Auto Range Clip to Range	Min Max 0 0
Draw Profiles Draw Mesh	Surfaces Filter Text
	substrate-wall 🔺
Coloring Banded	x=130 x=165
<ul> <li>Smooth</li> <li>Levels Setup</li> </ul>	x=95 E y-coordinate-9
20 🚔 1 🚔	y=0
	Display Compute Close Help

- a. Make sure that **Filled** and **Draw Mesh** are enabled in the **Options** group box.
- b. Disable **Global Range** in the **Options** group box.
- c. Select Velocity... and X Velocity from the Contours of drop-down lists.
- d. Select x=130, x=165, and x=95 in the Surfaces selection list, and deselect y=0.
- e. Click **Display** and close the **Contours** dialog box.

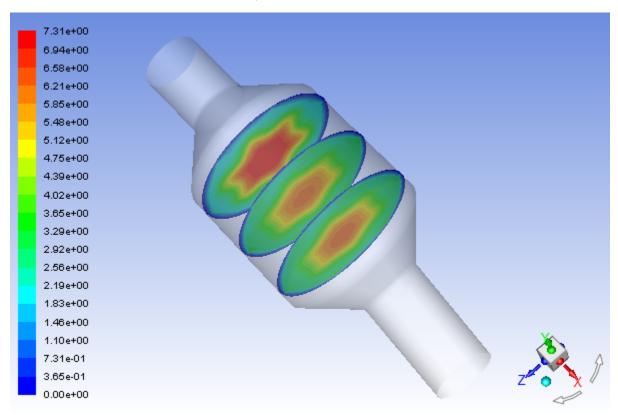


Figure 8.7: Contours of the X Velocity on the x=95, x=130, and x=165 Surfaces

The velocity profile becomes more uniform as the fluid passes through the porous media. The velocity is very high at the center (the area in red) just before the nitrogen enters the substrate and then decreases as it passes through and exits the substrate. The area in green, which corresponds to a moderate velocity, increases in extent.

11. Use numerical reports to determine the average, minimum, and maximum of the velocity distribution before and after the porous substrate.

Postprocessing  $\rightarrow$  Reports  $\rightarrow$  Surface Integrals...

Surface Integrals	
Report Type	Field Variable
Mass-Weighted Average 🔹	Velocity 🔻
Custom Vectors	X Velocity 👻
Vectors of	Surfaces Filter Text
Custom Vectors	porous-cl 🔹
	porous-in
Save Output Parameter	porous-out substrate-wall
	wall
	x=130
	x=165
	x=95 y-coordinate-9
	y=0
	Highlight Surfaces
	Mass-Weighted Average (m/s)
	4.677431
Compute	rite Close Help

- a. Select Mass-Weighted Average from the Report Type drop-down list.
- b. Select Velocity and X Velocity from the Field Variable drop-down lists.
- c. Select x=165 and x=95 in the Surfaces selection list.
- d. Click Compute.
- e. Select Facet Minimum from the Report Type drop-down list and click Compute.
- f. Select Facet Maximum from the Report Type drop-down list and click Compute.

The numerical report of average, maximum and minimum velocity can be seen in the main ANSYS Fluent console.

g. Close the Surface Integrals dialog box.

The spread between the average, maximum, and minimum values for X velocity gives the degree to which the velocity distribution is non-uniform. You can also use these numbers to calculate the velocity ratio (that is, the maximum velocity divided by the mean velocity) and the space velocity (that is, the product of the mean velocity and the substrate length).

Custom field functions and UDFs can be also used to calculate more complex measures of non-uniformity, such as the standard deviation and the gamma uniformity index.

Mass-Weighted Average X Velocity	(m/s)
x=95	5.3014566

x=165	4.0395238
Net	4.6774315
Minimum of Facet Values X Velocity	(m/s)
x=95 x=165	0.8805235 2.3446729
Net	0.8805235
Maximum of Facet Values X Velocity	(m/s)
x=95 x=165	8.0334597 6.3715858
Net	8.0334597

## 8.5. Summary

In this tutorial, you learned how to set up and solve a problem involving gas flow through porous media in ANSYS Fluent. You also learned how to perform appropriate postprocessing. Flow non-uniformities were easily identified through images of velocity vectors and pressure contours. Surface integrals and X-Y plots provided purely numeric data.

For additional details about modeling flow through porous media (including heat transfer and reaction modeling), see the Fluent User's Guide, available in the help viewer or on the ANSYS Customer Portal (http://support.ansys.com/documentation).

## **8.6. Further Improvements**

This tutorial guides you through the steps to reach an initial solution. You may be able to obtain a more accurate solution by using an appropriate higher-order discretization scheme and by adapting the mesh. Mesh adaption can also ensure that the solution is independent of the mesh. These steps are demonstrated in Introduction to Using ANSYS Fluent: Fluid Flow and Heat Transfer in a Mixing Elbow (p. 121).

# **Chapter 9: Using a Single Rotating Reference Frame**

This tutorial is divided into the following sections:

- 9.1. Introduction
- 9.2. Prerequisites
- 9.3. Problem Description
- 9.4. Setup and Solution
- 9.5. Summary
- 9.6. Further Improvements
- 9.7. References

# 9.1. Introduction

This tutorial considers the flow within a 2D, axisymmetric, co-rotating disk cavity system. Understanding the behavior of such flows is important in the design of secondary air passages for turbine disk cooling.

This tutorial demonstrates how to do the following:

- Set up a 2D axisymmetric model with swirl, using a rotating reference frame.
- Use the standard  $k \varepsilon$  and RNG  $k \varepsilon$  turbulence models.
- Calculate a solution using the pressure-based solver.
- Display velocity vectors and contours of pressure.
- Set up and display XY plots of radial velocity and wall  $y^+$  distribution.
- Restart the solver from an existing solution.

## 9.2. Prerequisites

This tutorial is written with the assumption that you have completed one or more of the introductory tutorials found in this manual:

- Introduction to Using ANSYS Fluent in ANSYS Workbench: Fluid Flow and Heat Transfer in a Mixing Elbow (p. 1)
- Parametric Analysis in ANSYS Workbench Using ANSYS Fluent (p. 73)
- Introduction to Using ANSYS Fluent: Fluid Flow and Heat Transfer in a Mixing Elbow (p. 121)

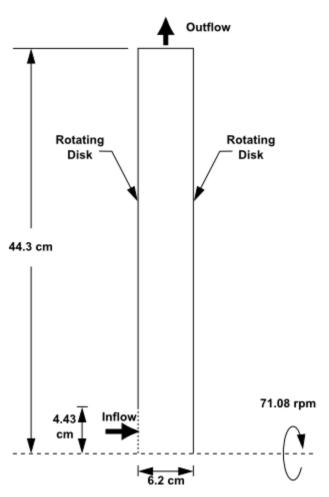
and that you are familiar with the ANSYS Fluent tree and ribbon structure. Some steps in the setup and solution procedure will not be shown explicitly.

# 9.3. Problem Description

The problem to be considered is shown schematically in Figure 9.1: Problem Specification (p. 394). This case is similar to a disk cavity configuration that was extensively studied by Pincombe [1].

Air enters the cavity between two co-rotating disks. The disks are 88.6 cm in diameter and the air enters at 1.146 m/s through a circular bore 8.86 cm in diameter. The disks, which are 6.2 cm apart, are spinning at 71.08 rpm, and the air enters with no swirl. As the flow is diverted radially, the rotation of the disk has a significant effect on the viscous flow developing along the surface of the disk.





As noted by Pincombe [1], there are two nondimensional parameters that characterize this type of disk cavity flow: the volume flow rate coefficient,  $C_w$ , and the rotational Reynolds number,  $Re_{\varphi}$ . These parameters are defined as follows:

$$C_w = \frac{Q}{v r_{out}} \tag{9.1}$$

$$Re_{\varphi} = \frac{\Omega r_{out}^2}{V} \tag{9.2}$$

where Q is the volumetric flow rate,  $\Omega$  is the rotational speed,  $\nu$  is the kinematic viscosity, and  $r_{out}$  is the outer radius of the disks. Here, you will consider a case for which  $C_w = 1092$  and  $Re_{\varphi} = 10^5$ .

# 9.4. Setup and Solution

The following sections describe the setup and solution steps for this tutorial:

9.4.1. Preparation 9.4.2. Mesh 9.4.3. General Settings 9.4.4. Models 9.4.5. Materials 9.4.6. Cell Zone Conditions 9.4.7. Boundary Conditions 9.4.8. Solution Using the Standard k-  $\varepsilon$  Model 9.4.9. Postprocessing for the Standard k-  $\varepsilon$  Solution 9.4.10. Solution Using the RNG k-  $\varepsilon$  Model 9.4.11. Postprocessing for the RNG k-  $\varepsilon$  Solution

### 9.4.1. Preparation

To prepare for running this tutorial:

- 1. Set up a working folder on the computer you will be using.
- 2. Go to the ANSYS Customer Portal, https://support.ansys.com/training.

#### Note

If you do not have a login, you can request one by clicking **Customer Registration** on the log in page.

- 3. Enter the name of this tutorial into the search bar.
- 4. Narrow the results by using the filter on the left side of the page.
  - a. Click ANSYS Fluent under Product.
  - b. Click **18.0** under **Version**.
- 5. Select this tutorial from the list.
- 6. Click the **single\_rotating\_R180.zip** link to download the input files.
- 7. Unzip single\_rotating\_R180.zip to your working folder.

The file disk.msh can be found in the single\_rotating folder created after unzipping the file.

8. Use Fluent Launcher to start the **2D** single precision (disable **Double Precision**) version of ANSYS Fluent.

Fluent Launcher displays your **Display Options** preferences from the previous session.

For more information about Fluent Launcher, see starting ANSYS Fluent using the Fluent Launcher in the Fluent Getting Started Guide.

9. Ensure that the Display Mesh After Reading and Workbench Color Scheme options are enabled.

### 10. Run in Serial under Processing Options.

## 9.4.2. Mesh

1. Read the mesh file (disk.msh).

File → Read → Mesh...

As ANSYS Fluent reads the mesh file, it will report its progress in the console.

#### Note

The Fluent console will display a warning that the current setup for the boundary conditions is not appropriate for a 2D/3D flow problem.

You will resolve this issue when you modify the solver settings in a subsequent step.

## 9.4.3. General Settings

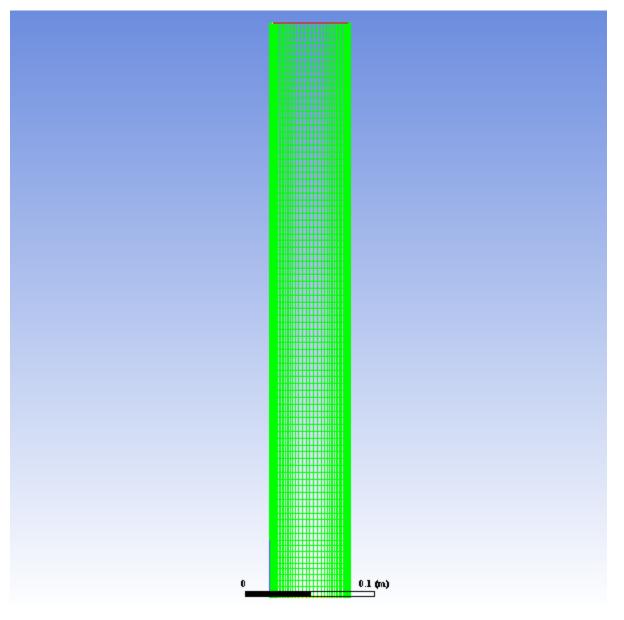
1. Check the mesh.

### **Setting Up Domain** $\rightarrow$ Mesh $\rightarrow$ Check

ANSYS Fluent will perform various checks on the mesh and report the progress in the console. Make sure that the reported minimum volume is a positive number.

2. Examine the mesh (Figure 9.2: Mesh Display for the Disk Cavity (p. 397)).





#### Extra

You can use the right mouse button to check which zone number corresponds to each boundary. If you click the right mouse button on one of the boundaries in the graphics window, information will be displayed in the ANSYS Fluent console about the associated zone, including the name of the zone. This feature is especially useful when you have several zones of the same type and you want to distinguish between them quickly.

3. Define new units for angular velocity and length.

### Setting Up Domain $\rightarrow$ Mesh $\rightarrow$ Units...

In the problem description, angular velocity and length are specified in rpm and cm, respectively, which is more convenient in this case. These are not the default units for these quantities.

Set Units			<b>X</b>
Quantities		Units	Set All to
heat-generation-rate heat-transfer-coefficient ignition-energy kinematic-viscosity	^	m cm mm in	default si british
length length-inverse length-time-inverse mag-permeability		ft	cgs
mass mass-diffusivity	New List	Factor 0.01 Offset 0 Close Help	
			±.

- a. Select angular-velocity from the Quantities list, and rpm in the Units list.
- b. Select length from the Quantities list, and cm in the Units list.
- c. Close the Set Units dialog box.

4. Specify the solver formulation to be used for the model calculation and enable the modeling of axisymmetric swirl.

Setting	g Up Physics $ ightarrow$ So	lver	
		Solver	
Time	Type	Velocity Formulation	Operating Conditions
<ul> <li>Steady</li> <li>Transient</li> </ul>	<ul> <li>Pressure-Based</li> <li>Density-Based</li> </ul>	<ul> <li>Absolute</li> <li>Relative</li> </ul>	🥏 Reference Values
	0	0	Axisymmetric Swirl

- a. Retain the default selection of **Pressure-Based** in the **Type** list.
- b. Retain the default selection of Absolute in the Velocity Formulation list.

For a rotating reference frame, the absolute velocity formulation has some numerical advantages.

c. Select Axisymmetric Swirl from the drop-down list in the Solver group box (below Reference Values...).

### 9.4.4. Models

1. Enable the standard  $k - \varepsilon$  turbulence model with the enhanced near-wall treatment.

Setting Up Physics  $\rightarrow$  Models  $\rightarrow$  Viscous...

Model       Model Constants         Inviscid       Inviscid         Laminar       0.09         Spalart-Allmaras (1 eqn)       0.09         k-epsilon (2 eqn)       1.44         Transition sST (4 eqn)       1.44         Reynolds Stress (7 eqn)       1.92         Scale-Adaptive Simulation (DES)       TKE Prandtl Number         k-epsilon Model       1.3         Standard       RNG         Realizable       User-Defined Functions         Non-Equilibrium Wall Functions       TKE Prandtl Number         Scalable Wall Functions       Turbulent Viscosity         Non-Equilibrium Wall Functions       TKE Prandtl Number         Menter-Lechner       Valuer-Defined Wall Treatment         User-Defined Wall Treatment Options       TDR Prandtl Number         Pressure Gradient Effects       TDR Prandtl Number	Viscous Model	×
	Model Inviscid Laminar Spalart-Allmaras (1 eqn) K-epsilon (2 eqn) K-epsilon (2 eqn) Transition k-kl-omega (3 eqn) Transition SST (4 eqn) Reynolds Stress (7 eqn) Scale-Adaptive Simulation (SAS) Detached Eddy Simulation (DES) K-epsilon Model Standard RNG Realizable Near-Wall Treatment Standard Wall Functions Scalable Wall Functions Enhanced Wall Treatment Menter-Lechner User-Defined Wall Functions	Model Constants Cmu 0.09 C1-Epsilon 1.44 C2-Epsilon 1.92 TKE Prandtl Number 1 TDR Prandtl Number 1.3 User-Defined Functions Turbulent Viscosity None Prandtl Numbers TKE Prandtl Number TKE Prandtl Number TDR Prandtl Number TDR Prandtl Number TDR Prandtl Number
<ul> <li>Production Kato-Launder</li> <li>Production Limiter</li> </ul>	Pressure Gradient Effects     Options     Production Kato-Launder	

a. Select k-epsilon (2 eqn) in the Model list.

The Viscous Model dialog box will expand.

- b. Retain the default selection of **Standard** in the **k-epsilon Model** list.
- c. Select Enhanced Wall Treatment in the Near-Wall Treatment list.
- d. Click **OK** to close the **Viscous Model** dialog box.

The ability to calculate a swirl velocity permits the use of a 2D mesh, so the calculation is simpler and more economical to run. This is especially important for problems where the enhanced wall treatment is used. The near-wall flow field is resolved through the viscous sublayer and buffer zones (that is, the first mesh point away from the wall is placed at a  $y^+$  of the order of 1).

For details, see enhanced wall treatment in the Fluent Theory Guide.

### 9.4.5. Materials

For the present analysis, you will model air as an incompressible fluid with a density of 1.225 kg/m<sup>3</sup> and a dynamic viscosity of  $1.7894 \times 10^{-5}$  kg/m-s. Since these are the default values, no change is required in the **Create/Edit Materials** dialog box.

1. Retain the default properties for air.

### Setting Up Physics $\rightarrow$ Materials $\rightarrow$ Create/Edit...

Create/Edit Materials			<b>—</b>				
Name	Material Type		Order Materials by				
air	fluid	•	<ul> <li>Name</li> <li>Chemical Formula</li> </ul>				
Chemical Formula	Fluent Fluid Materials	Fluent Fluid Materials					
	air	•	Fluent Database				
	Mixture						
	none	Ŧ	User-Defined Database				
Properties							
Density (kg/m3) constant	▼ Edit						
1.225							
Viscosity (kg/m-s) constant   Edit							
1.7894e-05							
Change/Create Delete Close Help							

#### Extra

You can modify the fluid properties for air at any time or copy another material from the database.

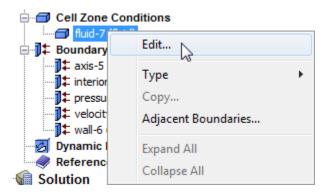
#### 2. Click Close to close the Create/Edit Materials dialog box.

For details, see physical properties in the Fluent User's Guide.

## 9.4.6. Cell Zone Conditions

Set up the present problem using a rotating reference frame for the fluid. Then define the disk walls to rotate with the moving frame.

**E** Setup  $\rightarrow$  Cell Zone Conditions  $\rightarrow$  fluid-7  $\stackrel{0}{\xrightarrow{}}$  Edit...



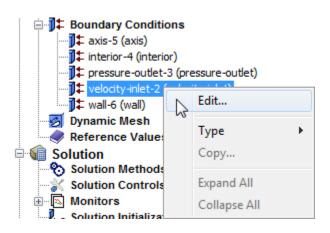
1. Define the rotating reference frame for the fluid zone (**fluid-7**).

E Fluid						<b>×</b>
Zone Name fluid-7						
Material Name air 🔻 Edit						
V Frame Motion 🗌 Laminar Zone 🗐 Source Terms						
Mesh Motion Fixed Values						
Porous Zone						
Reference Frame Mesh Motion Porous Zone 3D	Fan Zone	Embedded LES	Reaction	Source Terms	Fixed Values	Multiphase
Relative Specification UDF Relative To Cell Zone absolute  Zone Motion Function	none	•				
Rotational Velocity	- Translation	nal Velocity				
Speed (rpm) 71.08 constant •	X (m/s) 0	const	ant	-		
Copy To Mesh Motion	Y (m/s) 0	const	ant	-		
	ОК	Cancel Help				

- a. Enable Frame Motion.
- b. Enter 71.08 rpm for Speed in the Rotational Velocity group box.
- c. Click **OK** to close the **Fluid** dialog box.

### 9.4.7. Boundary Conditions

**E** Setup  $\rightarrow$  Boundary Conditions  $\rightarrow$  velocity-inlet-2  $\stackrel{\frown}{\Box}$  Edit...



1. Set the following conditions at the flow inlet (velocity-inlet-2).

💶 Velocity Inlet	:							×
Zone Name								
velocity-inlet-2								
Momentum	Thermal	Radiation	Species	DPM	Multipl	nase	Potential	UDS
Velocit	ty Specificatio	on Method Co	mponents					•
	Refere	nce Frame Ab	solute					
Supersonic/Init	Supersonic/Initial Gauge Pressure (pascal) 0 constant							
Axial-Velocity (m/s) 1.146 constant								
Radial-Velocity (m/s) 0 constant								
Swirl-Velocity (m/s) 0 constant								
Swirl Angular Velocity (rpm) 0								
Turbulence								
	Specificatio	n Method Inte	ensity and N	/iscosity	Ratio			
Turbulent Intensity (%) 5								
Turbulent Viscosity Ratio 5								
OK Cancel Help								

- a. Select Components from the Velocity Specification Method drop-down list.
- b. Enter 1.146 m/s for **Axial-Velocity**.
- c. Retain the default selection of **Intensity and Viscosity Ratio** from the **Specification Method** dropdown list in the **Turbulence** group box.
- d. Retain the default value of 5 % for **Turbulent Intensity**.
- e. Enter 5 for Turbulent Viscosity Ratio.
- f. Click **OK** to close the **Velocity Inlet** dialog box.

2. Set the following conditions at the flow outlet (**pressure-outlet-3**).

-	<b>الله</b>
Setup → Boundary Conditions → pressure-outlet-3	⊖ Edit

Pressure Outlet							
Zone Name							
pressure-outlet-3							
Momentum	Thermal	Radiation	Species	DPM	Multiphase	Potential	UDS
Back	Backflow Reference Frame Absolute						
	Gauge Pr	essure (paso	cal) 0		CC	onstant	
Backflow Direction	Backflow Direction Specification Method From Neighboring Cell				•		
Backflow	Pressure S	pecification	Total Pressu	ire			
Radial Equilibrium Pressure Distribution							
Average Press	Average Pressure Specification						
Target Mass Flow Rate							
	Turbulence						
	Specification Method Intensity and Viscosity Ratio						
Backflow Turbulent Intensity (%) 5			P				
Backflow Turbulent Viscosity Ratio 10							
L							
			Cance	Help	]		

- a. Select From Neighboring Cell from the Backflow Direction Specification Method drop-down list.
- b. Click **OK** to close the **Pressure Outlet** dialog box.

#### Note

ANSYS Fluent will use the backflow conditions only if the fluid is flowing into the computational domain through the outlet. Since backflow might occur at some point during the solution procedure, you should set reasonable backflow conditions to prevent convergence from being adversely affected.

3. Confirm that the wall boundary condition settings for the disk walls (wall-6) are as shown below.



🔁 Wall					<b>X</b>
Zone Name					
wall-6					
Adjacent Cell Zone fluid-7					
Momentum Thermal Radiation	Species DPM	Multiphase	UDS	Wall Film	Potential
Wall Motion Stationary Wall Moving Wall Shear Condition No Slip	ent Cell Zone				
<ul> <li>Specified Shear</li> <li>Specularity Coefficient</li> <li>Marangoni Stress</li> </ul>					
Wall Roughness					
Roughness Height (cm)	constant		-		
Roughness Constant 0.5	constant		<b>_</b>		
	OK Cancel	Help			

#### Note

A Stationary Wall condition implies that the wall is stationary with respect to the adjacent cell zone. Hence, in the case of a rotating reference frame a Stationary Wall is actually rotating with respect to the absolute reference frame. To specify a non-rotating wall in this case you would select Moving Wall (that is, moving with respect to the rotating reference frame). Then you would specify an absolute rotational speed of 0 in the Motion group box.

a. Click **OK** to close the **Wall** dialog box.

### 9.4.8. Solution Using the Standard k- $\epsilon$ Model

1. Set the solution parameters.



Solution Methods	
Pressure-Velocity Coupling	
Scheme	
Coupled 👻	
Spatial Discretization	
Pressure	
PRESTO!	
Momentum	
Second Order Upwind 🗸	
Swirl Velocity	
Second Order Upwind 👻	Ξ
Turbulent Kinetic Energy	
Second Order Upwind 🔻	
Turbulent Dissipation Rate	
Second Order Upwind 🔻	-
Transient Formulation	
· · · · · · · · · · · · · · · · · · ·	
Non-Iterative Time Advancement	
Frozen Flux Formulation	
Pseudo Transient	
Warped-Face Gradient Correction	
High Order Term Relaxation Options	
Default	

Help

- a. Select **Coupled** from the **Scheme** drop-down list in the **Pressure-Velocity Coupling** group box.
- b. Retain the default selection of Least Squares Cell Based from the Gradient list in the Spatial Discretization group box.
- c. Select **PRESTO!** from the **Pressure** drop-down list in the **Spatial Discretization** group box.

The PRESTO! scheme is well suited for steep pressure gradients involved in rotating flows. It provides improved pressure interpolation in situations where large body forces or strong pressure variations are present as in swirling flows.

d. Select **Second Order Upwind** from the **Turbulent Kinetic Energy** and **Turbulent Dissipation Rate** drop-down lists.

Use the scroll bar to access the discretization schemes that are not initially visible in the task page.

e. Enable **Pseudo Transient**.

The Pseudo Transient option enables the pseudo transient algorithm in the coupled pressure-based solver. This algorithm effectively adds an unsteady term to the solution equations in order to improve stability and convergence behavior. Use of this option is recommended for general fluid flow problems.

2. Set the solution controls.

Solv	/ing →	Controls	$\rightarrow$ (	Controls
------	--------	----------	-----------------	----------

Solution Controls	
Pseudo Transient Explicit Relaxation Factors	
Pressure	<b>.</b>
0.5	
Momentum	
0.5	Ξ
Density	
1	
Body Forces	
1	
Swirl Velocity	
0.75	-
Default	
Equations Limits Advanced	
Help	

a. Retain the default values in the Pseudo Transient Explicit Relaxation Factors group box.

#### Note

For this problem, the default explicit relaxation factors are satisfactory. However, if the solution diverges or the residuals display large oscillations, you may need to reduce the relaxation factors from their default values.

For tips on how to adjust the explicit relaxation parameters for different situations, see the Fluent User's Guide.

3. Enable the plotting of residuals during the calculation.



Residual Monitors					×
Options	Equations				
Print to Console	Residual	Monito	r Check Converge	nce Absolute Criteria	
V Plot	continuity	<b>v</b>	$\mathbf{\nabla}$	0.001	
Window	x-velocity	<b>V</b>		0.001	
1 Curves Axes	y-velocity	V	V	0.001	
Iterations to Plot	swirl	V		0.001	
1000 🚖	Residual Values	leral.		Convergence Criterio	
				Convergence Criterio	
Iterations to Store	Normalize		Iterations 5	absolute	
1000	Scale			Convergence Condi	tions
	Compute Loca	Scale			
OK Plot Renormalize Cancel Help					

- a. Ensure that **Plot** is enabled in the **Options** group box.
- b. Click OK to close the Residual Monitors dialog box.

#### Note

For this calculation, the convergence tolerance on the continuity equation is kept at 0.001. Depending on the behavior of the solution, you can reduce this value if necessary.

4. Enable the plotting of mass flow rate at the flow exit.

Solving  $\rightarrow$  Reports  $\rightarrow$  Definitions  $\rightarrow$  New  $\rightarrow$  Surface Report  $\rightarrow$  Mass Flow Rate

Surface Report Definition	
Name	Report Type
surf-mon-1	Mass Flow Rate
Options	Custom Vectors Vectors of
Per Surface	<b></b>
Average Over	Custom Vectors
	Field Variable
Report Files [0/0]	Pressure
	Static Pressure 👻
	Surfaces Filter Text
	axis-5
	interior-4
Report Plots [0/0]	
	velocity-inlet-2 wall-6
	won-o
Create	
Report File	
Report Plot	
Frequency 1	
Print to Console	
Create Output Parameter	New Surface
ОК	Compute Cancel Help

- a. Enter **surf-mon-1** for the **Name** of the surface report definition.
- b. In the Create group box, enable Report File, Report Plot and Print to Console for surf-mon-1.

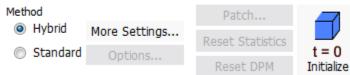
#### Note

When the **Report File** option is selected in the **Surface Report Definition** dialog box, the mass flow rate history will be written to a file. If you do not enable the **Report File** option, the history information will be lost when you exit ANSYS Fluent.

- c. Select **pressure-outlet-3** from the **Surfaces** selection list.
- d. Click **OK** to save the surface report definition settings and close the **Surface Report Definition** dialog box.
- 5. Initialize the solution.

Solving → Initialization

#### Initialization



- a. Retain the default selection of the **Hybrid** initialization method.
- b. Click Initialize.

#### Note

For flows in complex topologies, hybrid initialization will provide better initial velocity and pressure fields than standard initialization. This in general will help in improving the convergence behavior of the solver.

6. Save the case file (disk-ke.cas.gz).



7. Start the calculation by requesting 600 iterations.

# Run Calculation

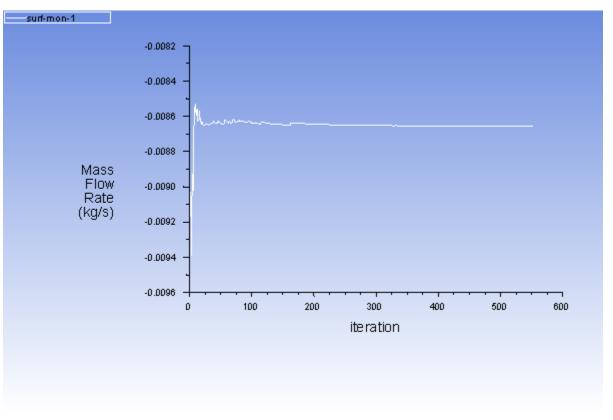
Update Dynamic Mesh	No. of Iterations	600		_ ',
Input Summary	No. of Icelacons	000	•	-⁄-
Advanced	Check Case			Calculate

a. Enter 600 for the Number of Iterations.

#### b. Click Calculate.

Throughout the calculation, ANSYS Fluent will report reversed flow at the exit. This is reasonable for the current case. The solution should be sufficiently converged after approximately 550 iterations. The mass flow rate history is shown in Figure 9.3: Mass Flow Rate History (k- $\varepsilon$  Turbulence Model) (p. 410).





#### Extra

Here we have retained the default **Timescale Factor** of 1 in the **Run Calculation** panel. When performing a Pseudo Transient calculation, larger values of **Timescale Factor** may speed up convergence of the solution. However, setting **Timescale Factor** too large may cause the solution to diverge and fail to complete. As an optional activity, you can reinitialize the solution and try running the calculation with **Timescale Factor** set to 2. Observe the convergence behavior and the number of iterations before convergence. Then try the same again with **Timescale Factor** set to 4. For more information on setting Timescale Factor and the Pseudo Transient solver settings, refer to the Fluent User's Guide.

8. Check the mass flux balance.

### Postprocessing ightarrow Reports ightarrow Fluxes...

#### Warning

Although the mass flow rate history indicates that the solution is converged, you should also check the net mass fluxes through the domain to ensure that mass is being conserved.

E Flux Reports		
Options Mass Flow Rate	Boundaries Filter Text	Results
<ul> <li>Total Heat Transfer Rate</li> <li>Radiation Heat Transfer Rate</li> </ul>	axis-5 interior-4	
	pressure-outlet-3 velocity-inlet-2	-0.008655054494738579 0.008655219338834286
	wall-6	
	<	
Save Output Parameter	2 L	Net Results (kg/s)
		1.648441e-07
	Compute Write Close Help	

- a. Select velocity-inlet-2 and pressure-outlet-3 from the Boundaries selection list.
- b. Retain the default Mass Flow Rate option.
- c. Click **Compute** and close the **Flux Reports** dialog box.

#### Warning

The net mass imbalance should be a small fraction (for example, 0.5%) of the total flux through the system. If a significant imbalance occurs, you should decrease the residual tolerances by at least an order of magnitude and continue iterating.

9. Save the case and data files (disk-ke.cas.gz and disk-ke.dat.gz).

**File**  $\rightarrow$  Write  $\rightarrow$  Case & Data...

Click OK to overwrite disk-ke.cas.gz.

#### Note

It is always prudent to save both case and data files in case anything has changed.

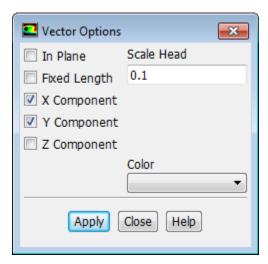
### 9.4.9. Postprocessing for the Standard k- $\epsilon$ Solution

1. Display the velocity vectors.

Postprocessing  $\rightarrow$  Graphics  $\rightarrow$  Vectors  $\rightarrow$  Edit...

Vectors	
Options Global Range Auto Range Clip to Range Auto Scale Draw Mesh Style arrow Scale Skip	Vectors of Velocity Color by Velocity Velocity Magnitude Min Max 0 Surfaces Filter Text
Scale   Skip     50   1     Vector Options     Custom Vectors	axis-5 interior-4 pressure-outlet-3 velocity-inlet-2 wall-6 New Surface Display Compute Close Help

- a. Enter 50 for Scale
- b. Set Skip to 1.
- c. Click the Vector Options... button to open the Vector Options dialog box.

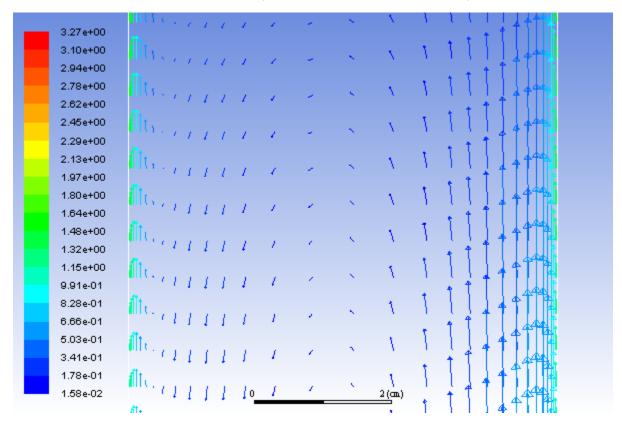


i. Disable **Z Component**.

This allows you to examine only the non-swirling components.

- ii. Click **Apply** and close the **Vector Options** dialog box.
- d. Click **Display** in the **Vectors** dialog box to plot the velocity vectors.

A magnified view of the velocity field displaying a counter-clockwise circulation of the flow is shown in Figure 9.4: Magnified View of Velocity Vectors within the Disk Cavity (p. 413).





- e. Close the **Vectors** dialog box.
- 2. Display filled contours of static pressure.

Postprocessing  $\rightarrow$  Graphics  $\rightarrow$  Contours  $\rightarrow$  Edit...

Contours	
Options Filled	Contours of Pressure
<ul> <li>Node Values</li> <li>Global Range</li> </ul>	Static Pressure
Auto Range Clip to Range	Min (pascal) Max (pascal) 0 0
Draw Profiles	Surfaces Filter Text
Coloring Banded Smooth	axis-5 interior-4 pressure-outlet-3 velocity-inlet-2 wall-6
Levels Setup 20 🐳 1 👻	New Surface 🔻
	Display Compute Close Help

a. Enable **Filled** in the **Options** group box.

b. Retain the selection of Pressure... and Static Pressure from the Contours of drop-down lists.

c. Click **Display** and close the **Contours** dialog box.

The pressure contours are displayed in Figure 9.5: Contours of Static Pressure for the Entire Disk Cavity (p. 415). Notice the high pressure that occurs on the right disk near the hub due to the stagnation of the flow entering from the bore.

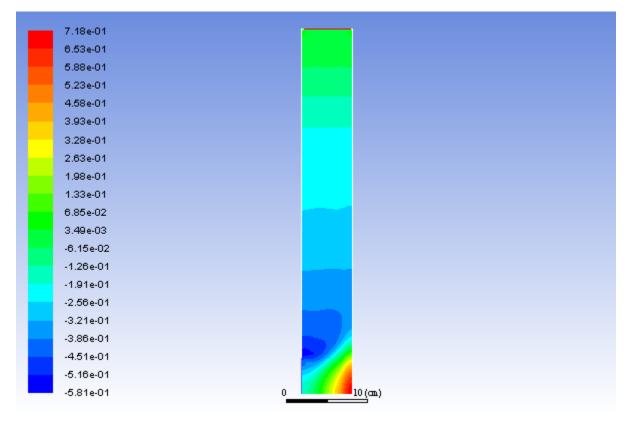


Figure 9.5: Contours of Static Pressure for the Entire Disk Cavity

3. Create a constant *y*-coordinate line for postprocessing.

Postprocessing  $\rightarrow$  Surface  $\rightarrow$  Create  $\rightarrow$  Iso-Surface...

Iso-Surface	
Surface of Constant	From Surface Filter Text
Mesh •	
Y-Coordinate -	axis-5 interior-4
Min Max	pressure-outlet-3
0 0	velocity-inlet-2
Iso-Values	wall-6
37	
↓ ] ▶	From Zones Fiter Text
New Surface Name	fluid-7
aaa_y=37cm	
Create	Manage Close Help

- a. Select Mesh... and Y-Coordinate from the Surface of Constant drop-down lists.
- b. Click **Compute** to update the minimum and maximum values.

c. Enter 37 in the Iso-Values field.

This is the radial position along which you will plot the radial velocity profile.

d. Enter aaa\_y=37cm for the **New Surface Name**.

Using a prefix such as **aaa** or **zzz** allows you to keep all postprocessing surfaces together.

e. Click Create to create the isosurface.

### Note

The name you use for an isosurface can be any continuous string of characters (without spaces).

- f. Close the **Iso-Surface** dialog box.
- 4. Plot the radial velocity distribution on the surface **aaa\_y=37cm**.

$\rightarrow$ Fostprocessing $\rightarrow$ Flots $\rightarrow$ AT Flot $\rightarrow$ Eult.	processing $ ightarrow$ Plots $ ightarrow$ XY Plot $ ightarrow$	P	10000	
--	---	---	-------	--

Solution XY Plot		<b>—</b>
Options Vode Values Position on X Axis Position on Y Axis Write to File Order Points File Data	Plot Direction X 1 Y 0 Z 0	Y Axis Function Velocity Radial Velocity X Axis Function Direction Vector Surfaces Fiter Text  aaa_y=37cm axis-5 interior-4 pressure-outlet-3 velocity-inlet-2 wall-6 New Surface
Plot	Axes Curves	Close Help

- a. Select Velocity... and Radial Velocity from the Y Axis Function drop-down lists.
- b. Select the y-coordinate line **aaa\_y=37cm** from the **Surfaces** selection list.
- c. Click Plot.

Figure 9.6: Radial Velocity Distribution—Standard k-  $\varepsilon$  Solution(p. 417) shows a plot of the radial velocity distribution along y=37cm.

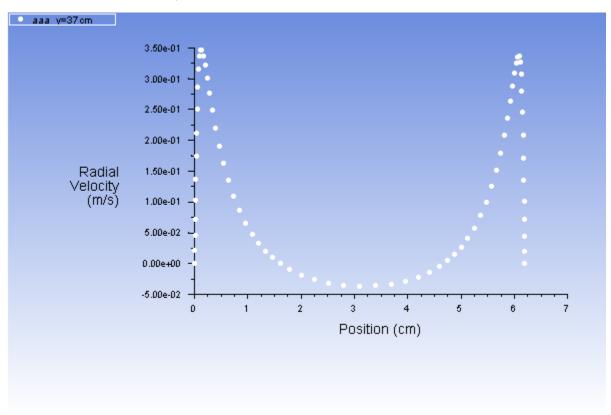


Figure 9.6: Radial Velocity Distribution—Standard k-  $\varepsilon$  Solution

- d. Enable Write to File in the Options group box to save the radial velocity profile.
- e. Click the Write... button to open the Select File dialog box.
  - i. Enter ke-data.xy in the XY File text entry box and click OK.

Be sure to double check the location where the files are being saved to ensure they will be saved where you intend.

5. Plot the wall y+ distribution on the rotating disk wall along the radial direction (Figure 9.7: Wall Yplus Distribution on wall-6— Standard k- ε Solution (p. 420)).

Postprocessing  $\rightarrow$  Plots  $\rightarrow$  XY Plot  $\rightarrow$  Edit...

Solution XY Plot			
Options Vode Values Position on X Axis Position on Y Axis Vrite to File Order Points File Data		Plot Direction X 0 Y 1 Z 0 Load File Free Data	Y Axis Function Turbulence Wall Yplus X Axis Function Direction Vector Surfaces Filter Text aaa_y=37cm axis-5 interior-4 pressure-outlet-3 velocity-inlet-2 wall-6 New Surface ▼
	Plot	Axes Curves.	Close Help

- a. Disable Write to File in the Options group box.
- b. Select Turbulence... and Wall Yplus from the Y Axis Function drop-down lists.
- c. Deselect aaa\_y=37cm and select wall-6 from the Surfaces selection list.
- d. Enter 0 and 1 for **X** and **Y** respectively in the **Plot Direction** group box.

### Note

The change in Plot Direction is required because we are plotting y+ along the radial dimension of the disk, which is oriented with Y axis.

e. Click the Axes... button to open the Axes - Solution XY Plot dialog box.

Axes - Solution XY Plot			×
Axis X Y Label	Number Format Type general Precision 3	Major Rules Color foreground Weight 1	•
Options Log Auto Range Major Rules Minor Rules	Range Minimum 0 Maximum 43	Minor Rules Color dark gray Weight 1	•
Appl	y Close Help		

- i. Retain the default selection of **X** from the **Axis** group box.
- ii. Disable Auto Range in the Options group box.
- iii. Retain the default value of 0 for **Minimum** and enter 43 for **Maximum** in the **Range** group box.
- iv. Click Apply and close the Axes Solution XY Plot dialog box.
- f. Click **Plot** in the **Solution XY Plot** dialog box.

Figure 9.7: Wall Yplus Distribution on wall-6— Standard k-  $\varepsilon$  Solution(p. 420) shows a plot of wall y+ distribution along **wall-6**.

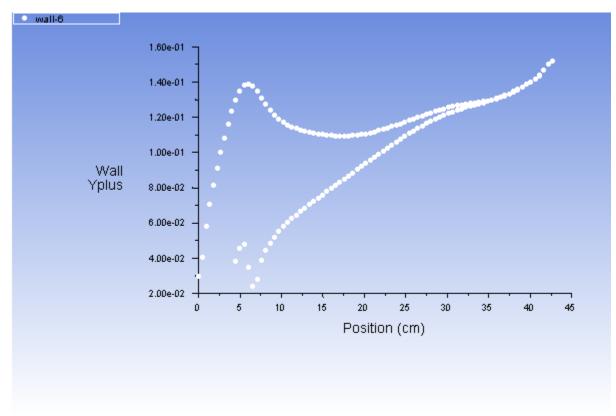


Figure 9.7: Wall Yplus Distribution on wall-6— Standard k-  $\epsilon$  Solution

- g. Enable Write to File in the Options group box to save the wall y+ profile.
- h. Click the Write... button to open the Select File dialog box.
  - i. Enter ke-yplus.xy in the **XY File** text entry box and click **OK**.

#### Note

Ideally, while using enhanced wall treatment, the wall y+ should be in the order of 1 (at least less than 5) to resolve the viscous sublayer. The plot justifies the applicability of enhanced wall treatment to the given mesh.

i. Close the Solution XY Plot dialog box.

# 9.4.10. Solution Using the RNG k- $\epsilon$ Model

Recalculate the solution using the RNG k- $\varepsilon$  turbulence model.

1. Enable the RNG k- $\varepsilon$  turbulence model with the enhanced near-wall treatment.



Viscous Model	<b>X</b>		
Model Inviscid Laminar Spalart-Allmaras (1 eqn) k-epsilon (2 eqn) k-omega (2 eqn) Transition k-kl-omega (3 eqn) Transition SST (4 eqn) Reynolds Stress (7 eqn) Scale-Adaptive Simulation (SAS) Detached Eddy Simulation (DES) k-epsilon Model	Model Constants Cmu 0.0845 C1-Epsilon 1.42 C2-Epsilon 1.68 Swirl Factor 0.07		
<ul> <li>Standard</li> <li>RNG</li> <li>Realizable</li> <li>RNG Options</li> <li>Differential Viscosity Model</li> <li>Swirl Dominated Flow</li> </ul>			
Near-Wall Treatment Standard Wall Functions Scalable Wall Functions Non-Equilibrium Wall Functions Enhanced Wall Treatment Menter-Lechner User-Defined Wall Functions			
Enhanced Wall Treatment Options  Pressure Gradient Effects			
Options <ul> <li>Production Kato-Launder</li> <li>Production Limiter</li> </ul>	User-Defined Functions Turbulent Viscosity none		
OK Cancel Help			

- a. Select **RNG** in the **k-epsilon Model** list.
- b. Enable Differential Viscosity Model and Swirl Dominated Flow in the RNG Options group box.

The differential viscosity model and swirl modification can provide better accuracy for swirling flows such as the disk cavity.

For more information, see RNG swirl modification in the Fluent Theory Guide.

#### c. Retain Enhanced Wall Treatment as the Near-Wall Treatment.

d. Click **OK** to close the **Viscous Model** dialog box.

2. Continue the calculation by requesting 300 iterations.

# **Solving** $\rightarrow$ Run Calculation

The solution converges after approximately 245 additional iterations.

3. Save the case and data files (disk-rng.cas.gz and disk-rng.dat.gz).

**File**  $\rightarrow$  Write  $\rightarrow$  Case & Data...

# 9.4.11. Postprocessing for the RNG k- $\epsilon$ Solution

1. Plot the radial velocity distribution for the RNG k- $\varepsilon$  solution and compare it with the distribution for the standard k- $\varepsilon$  solution.

10001 H000	Postprocessing $\rightarrow$ Plots $\rightarrow$ XY Plot $\rightarrow$ Edit	

Solution XY Plot			
Options <ul> <li>Options</li> <li>Node Values</li> <li>Position on X Axis</li> <li>Position on Y Axis</li> <li>Write to File</li> <li>Order Points</li> </ul> File Data [1/1] File Data [1/1] Radial Velocity	Plot Direction X 1 Y 0 Z 0	Y Axis Function Velocity Radial Velocity X Axis Function Direction Vector Surfaces Filter Text To T	
Plot Axes Curves Close Help			

- a. Enter 1 and 0 for  $\boldsymbol{X}$  and  $\boldsymbol{Y}$  respectively in the **Plot Direction** group box.
- b. Select Velocity... and Radial Velocity from the Y Axis Function drop-down lists.
- c. Select **aaa\_y=37cm** and deselect **wall-6** from the **Surfaces** selection list.
- d. Disable the Write to File option.
- e. Click the **Load File...** button to load the k-  $\varepsilon$  data.
  - i. Select the file **ke-data.xy** in the **Select File** dialog box.

- ii. Click **OK**.
- f. Click the **Axes...** button to open the **Axes Solution XY Plot** dialog box.
  - i. Enable Auto Range in the Options group box.
  - ii. Click Apply and close the Axes Solution XY Plot dialog box.
- g. Click the **Curves...** button to open the **Curves Solution XY Plot** dialog box, where you will define a different curve symbol for the RNG k- $\varepsilon$  data.

Curves - Solution XY Plot	<b>×</b>
Curve # 0 Sample Color foreground Weight 1 Apply Close	Marker Style Symbol x Color foreground Size 0.3

- i. Retain 0 for the Curve #.
- ii. Select **x** from the **Symbol** drop-down list.
- iii. Click Apply and close the Curves Solution XY Plot dialog box.
- h. Click **Plot** in the **Solution XY Plot** dialog box (Figure 9.8: Radial Velocity Distribution RNG k-  $\varepsilon$  and Standard k-  $\varepsilon$  Solutions (p. 424)).

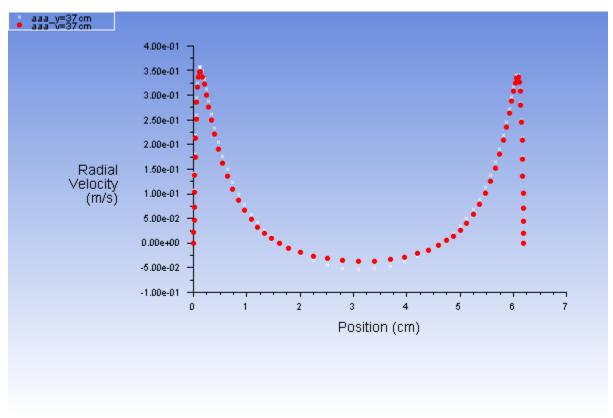


Figure 9.8: Radial Velocity Distribution — RNG k-  $\epsilon$  and Standard k-  $\epsilon$  Solutions

The peak velocity predicted by the RNG  $k - \varepsilon$  solution is higher than that predicted by the standard  $k - \varepsilon$  solution. This is due to the less diffusive character of the RNG  $k - \varepsilon$  model. Adjust the range of the x axis to magnify the region of the peaks.

i. Click the **Axes...** button to open the **Axes - Solution XY Plot** dialog box, where you will specify the *x*-axis range.

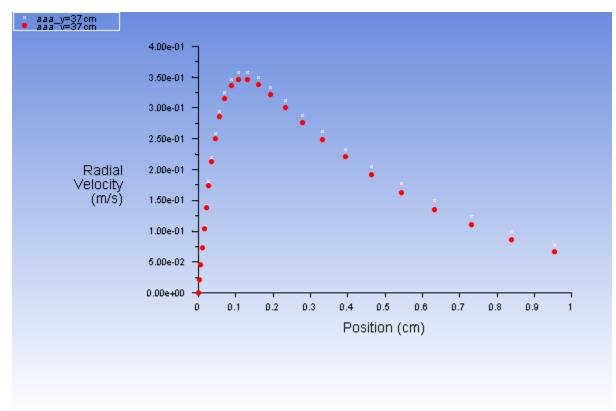
Axes - Solution XY Plot		×
Axis X Y Label	Number Format Type general • Precision 3 •	Major Rules Color foreground Weight 1
Options Log Auto Range Major Rules Minor Rules	Range Minimum 0 Maximum 1 Close Help	Minor Rules Color dark gray Weight 1

i. Disable Auto Range in the Options group box.

- ii. Retain the value of 0 for **Minimum** and enter 1 for **Maximum** in the **Range** dialog box.
- iii. Click Apply and close the Axes Solution XY Plot dialog box.
- j. Click **Plot**.

The difference between the peak values calculated by the two models is now more apparent.

Figure 9.9: RNG k-  $\varepsilon$  and Standard k-  $\varepsilon$  Solutions (x=0 cm to x=1 cm)



2. Plot the wall y+ distribution on the rotating disk wall along the radial direction Figure 9.10: wall-6 — RNG k- ε and Standard k- ε Solutions (x=0 cm to x=43 cm) (p. 427).



Vode Values   Position on X Axis   Position on Y Axis   Write to File   Order Points     File Data [1/1]     Load File   Wall Yplus   Free Data     Load File   Free Data     Image: Application on Y Axis     X 1   Y 0   Z 0   X Axis Function   Direction Vector     Surfaces Filter Text     aaa_y=37cm   axis-5   interior-4	
Plot Axes Curves Close Help	

- a. Select Turbulence... and Wall Yplus from the Y Axis Function drop-down lists.
- b. Deselect **aaa\_y=37cm** and select **wall-6** from the **Surfaces** selection list.
- c. Enter 0 and 1 for **X** and **Y** respectively in the **Plot Direction** group box.
- d. Select any existing files that appear in the **File Data** selection list and click the **Free Data** button to remove the file.
- e. Click the **Load File...** button to load the RNG k- $\varepsilon$  data.
  - i. Select the file ke-yplus.xy in the Select File dialog box.
  - ii. Click **OK**.
- f. Click the Axes... button to open the Axes Solution XY Plot dialog box.
  - i. Retain the default selection of **X** from the **Axis** group box.
  - ii. Retain the default value of 0 for **Minimum** and enter 43 for **Maximum** in the **Range** group box.
  - iii. Click **Apply** and close the **Axes Solution XY Plot** dialog box.
- g. Click **Plot** in the **Solution XY Plot** dialog box.

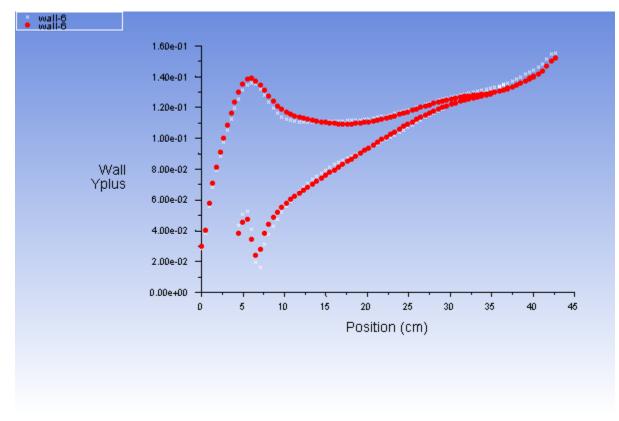


Figure 9.10: wall-6 — RNG k-  $\epsilon$  and Standard k-  $\epsilon$  Solutions (x=0 cm to x=43 cm)

# 9.5. Summary

This tutorial illustrated the setup and solution of a 2D, axisymmetric disk cavity problem in ANSYS Fluent. The ability to calculate a swirl velocity permits the use of a 2D mesh, thereby making the calculation simpler and more economical to run than a 3D model. This can be important for problems where the enhanced wall treatment is used, and the near-wall flow field is resolved using a fine mesh (the first mesh point away from the wall being placed at a y+ on the order of 1).

For more information about mesh considerations for turbulence modeling, see the Fluent User's Guide.

# 9.6. Further Improvements

The case modeled in this tutorial lends itself to parametric study due to its relatively small size. Here are some things you may want to try:

• Separate **wall-6** into two walls.

### Setting Up Domain $\rightarrow$ Zones $\rightarrow$ Separate $\rightarrow$ Faces...

Specify one wall to be stationary, and rerun the calculation.

• Use adaption to see if resolving the high velocity and pressure-gradient region of the flow has a significant effect on the solution.

- Introduce a nonzero swirl at the inlet or use a velocity profile for fully-developed pipe flow. This is probably more realistic than the constant axial velocity used here, since the flow at the inlet is typically being supplied by a pipe.
- Model compressible flow (using the ideal gas law for density) rather than assuming incompressible flow text.

This tutorial guides you through the steps to reach an initial solution. You may be able to obtain a more accurate solution by using an appropriate higher-order discretization scheme and by adapting the mesh. Mesh adaption can also ensure that the solution is independent of the mesh. These steps are demonstrated in Introduction to Using ANSYS Fluent: Fluid Flow and Heat Transfer in a Mixing Elbow (p. 121).

# 9.7. References

1. Pincombe, J.R., "Velocity Measurements in the Mk II - Rotating Cavity Rig with a Radial Outflow," Thermo-Fluid Mechanics Research Centre, University of Sussex, Brighton, UK, 1981.

# **Chapter 10: Using Multiple Reference Frames**

This tutorial is divided into the following sections:

10.1. Introduction
10.2. Prerequisites
10.3. Problem Description
10.4. Setup and Solution
10.5. Summary
10.6. Further Improvements

# 10.1. Introduction

Many engineering problems involve rotating flow domains. One example is the centrifugal blower unit that is typically used in automotive climate control systems. For problems where all the moving parts (fan blades, hub and shaft surfaces, and so on) are rotating at a prescribed angular velocity, and the stationary walls (for example, shrouds, duct walls) are surfaces of revolution with respect to the axis of rotation, the entire domain can be referred to as a single rotating frame of reference. However, when each of the several parts is rotating about a different axis of rotation, or about the same axis at different speeds, or when the stationary walls are not surfaces of revolution (such as the volute around a centrifugal blower wheel), a single rotating coordinate system is not sufficient to "immobilize" the computational domain so as to predict a steady-state flow field. In such cases, the problem must be formulated using multiple reference frames.

In ANSYS Fluent, the flow features associated with one or more rotating parts can be analyzed using the multiple reference frame (MRF) capability. This model is powerful in that multiple rotating reference frames can be included in a single domain. The resulting flow field is representative of a snapshot of the transient flow field in which the rotating parts are moving. However, in many cases the interface can be chosen in such a way that the flow field at this location is independent of the orientation of the moving parts. That is, if an interface can be drawn on which there is little or no angular dependence, the model can be a reliable tool for simulating time-averaged flow fields. It is therefore very useful in complicated situations where one or more rotating parts are present.

This tutorial illustrates the procedure for setting up and solving a problem using the MRF capability. As an example, the flow field on a 2D section of a centrifugal blower will be calculated. Although this is a general methodology that can be applied to cases where more than one reference frame is moving, this example will be limited to a single rotating reference frame.

This tutorial demonstrates how to do the following:

- · Create mesh interfaces from interface-zones defined during meshing.
- Specify different frames of reference for different fluid zones.
- Set the relative velocity of each wall.
- Calculate a solution using the pressure-based solver.

# 10.2. Prerequisites

This tutorial is written with the assumption that you have completed one or more of the introductory tutorials found in this manual:

- Introduction to Using ANSYS Fluent in ANSYS Workbench: Fluid Flow and Heat Transfer in a Mixing Elbow (p. 1)
- Parametric Analysis in ANSYS Workbench Using ANSYS Fluent (p. 73)
- Introduction to Using ANSYS Fluent: Fluid Flow and Heat Transfer in a Mixing Elbow (p. 121)

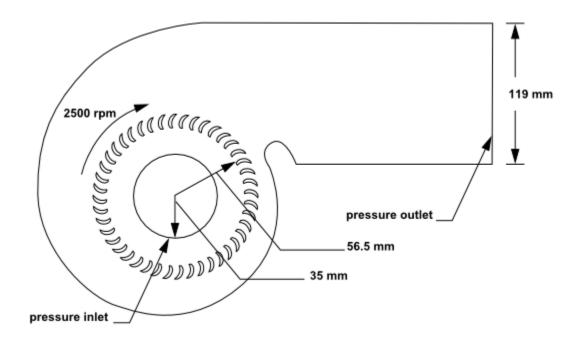
This tutorial also assumes that you are familiar with the ANSYS Fluent tree and ribbon structure. Some steps in the setup and solution procedure will not be shown explicitly.

In general, to solve problems using the MRF feature, you should be familiar with the concept of creating multiple fluid zones in your mesh generator.

# **10.3. Problem Description**

This problem considers a 2D section of a generic centrifugal blower. A schematic of the problem is shown in Figure 10.1: Schematic of the Problem (p. 430). The blower consists of 32 blades, each with a chord length of 13.5 mm. The blades are located approximately 56.5 mm (measured from the leading edge) from the center of rotation. The radius of the outer wall varies logarithmically from 80 mm to 146.5 mm. You will simulate the flow under no load, or free-delivery conditions when inlet and outlet pressures are at ambient conditions (0 Pa gauge). This corresponds to the maximum flow-rate of the blower when sitting in free air. The blades are rotating with an angular velocity of 2500 rpm. The flow is assumed to be turbulent.

#### Figure 10.1: Schematic of the Problem



# 10.4. Setup and Solution

The following sections describe the setup and solution steps for this tutorial:

- 10.4.1. Preparation
- 10.4.2. Reading and Checking the Mesh and Setting the Units
- 10.4.3. Specifying Solver and Analysis Type
- 10.4.4. Specifying the Models
- 10.4.5. Specifying Materials
- 10.4.6. Specifying Cell Zone Conditions
- 10.4.7. Setting Boundary Conditions
- 10.4.8. Defining Mesh Interfaces
- 10.4.9. Obtaining the Solution
- 10.4.10. Step 9: Postprocessing

# 10.4.1. Preparation

To access tutorials and their input files on the ANSYS Customer Portal, go to http://support.ansys.com/ training.

To prepare for running this tutorial:

- 1. Set up a working folder on the computer you will be using.
- 2. Go to the ANSYS Customer Portal, https://support.ansys.com/training.

### Note

If you do not have a login, you can request one by clicking **Customer Registration** on the log in page.

- 3. Enter the name of this tutorial into the search bar.
- 4. Narrow the results by using the filter on the left side of the page.
  - a. Click ANSYS Fluent under Product.
  - b. Click **18.0** under **Version**.
- 5. Select this tutorial from the list.
- 6. Click the **multiple\_rotating\_R180.zip** link to download the input files.
- 7. Unzip the multiple\_rotating\_R180.zip file you have downloaded to your working folder.

The file, blower-2d.msh can be found in the multiple\_rotating directory created after unzipping the file.

8. Use Fluent Launcher to start the **2D Double Precision** version of ANSYS Fluent.

Fluent Launcher displays your **Display Options** preferences from the previous session.

For more information about Fluent Launcher, see starting ANSYS Fluent using the Fluent Launcher in the Fluent Getting Started Guide.

- 9. Ensure that the **Display Mesh After Reading** and **Workbench Color Scheme** options are enabled.
- 10. Ensure that the **Serial** processing option is selected.

# 10.4.2. Reading and Checking the Mesh and Setting the Units

1. Read the mesh file (blower-2d.msh).

### File $\rightarrow$ Read $\rightarrow$ Mesh...

The geometry and mesh are displayed in graphics window (Figure 10.2: Mesh of the 2D Centrifugal Blower (p. 433))

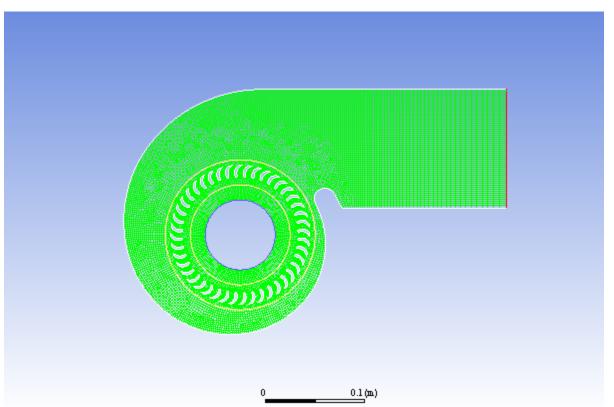
2. Check the mesh.

### **Setting Up Domain** $\rightarrow$ Mesh $\rightarrow$ Check

ANSYS Fluent will perform various checks on the mesh and will report the progress in the console. Make sure that the reported minimum volume is a positive number. It will also issue warnings about unassigned interface zones. You do not need to take any action now. You will set up the mesh interfaces in a later step.

3. Examine the mesh.

The mesh consists of three fluid zones, **fluid-casing**, **fluid-inlet**, and **fluid-rotor**. These are reported in the console when the mesh is read. In the **Mesh Display** dialog box, the fluid zones are reported as interior zones **default-interior**, **default-interior:013**, and **default-interior:015** respectively. The fluid zone containing the blades will be solved in a rotational reference frame.



The fluid zones are bounded by interface zones that appear in the mesh display in yellow. These interface boundaries were used in the mesh generator to separate the fluid zones, and will be used to create mesh interfaces between adjacent fluid zones when the boundary conditions are set later in this tutorial.

4. Set the units for angular velocity.

### Setting Up Domain $\rightarrow$ Mesh $\rightarrow$ Units...

In the problem description, angular velocity is specified in rpm rather than in the default unit of rad/s.

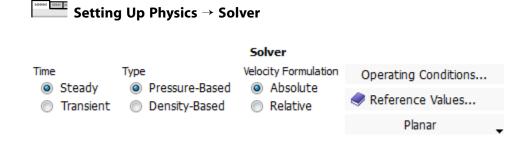
Set Units			<b>X</b>
Quantities		Units	Set All to
acceleration angle angular-velocity area area-inverse collision-rate concentration contact-resistance crank-angle crank-angular-velocity		rad/s deg/s rpm Factor 0.1047198	default si british cgs
	New List	Offset 0 Close Help	A

### Figure 10.2: Mesh of the 2D Centrifugal Blower

- a. Select angular-velocity from the Quantities list and rpm in the Units list.
- b. Close the Set Units dialog box.

# 10.4.3. Specifying Solver and Analysis Type

1. Retain the default settings of the pressure-based steady-state solver in the **Solver** group.



# 10.4.4. Specifying the Models

1. Enable the standard  $k - \varepsilon$  turbulence model.



<b>E</b> Viscous Model	<b>X</b>
Model	Model Constants
Inviscid	Cmu
C Laminar	0.09
Spalart-Allmaras (1 eqn)	C1-Epsilon
k-epsilon (2 eqn)	1.44
<ul> <li>k-omega (2 eqn)</li> <li>Transition k-kl-omega (3 eqn)</li> </ul>	C2-Epsilon
<ul> <li>Transition SST (4 eqn)</li> </ul>	1.92
Reynolds Stress (5 eqn)	TKE Prandtl Number
Scale-Adaptive Simulation (SAS)	1
Detached Eddy Simulation (DES)	TDR Prandtl Number
k-epsilon Model	1.3
Standard	
© RNG	
Realizable	
Near-Wall Treatment	User-Defined Functions
Standard Wall Functions	Turbulent Viscosity
Scalable Wall Functions	none
Non-Equilibrium Wall Functions	Prandtl Numbers
Enhanced Wall Treatment	TKE Prandtl Number
<ul> <li>Menter-Lechner</li> <li>User-Defined Wall Functions</li> </ul>	none
	TDR Prandtl Number
Enhanced Wall Treatment Options	none
Ontions	
Options Curvature Correction	
Production Kato-Launder	
Production Limiter	
OK C	Cancel Help

- a. Select **k-epsilon (2eqn)** in the **Model** list.
- b. Select Enhanced Wall Treatment in the Near-Wall Treatment list.
- c. Click **OK** to close the **Viscous Model** dialog box.

# 10.4.5. Specifying Materials

1. Retain the default properties for air.

**E** Setup 
$$\rightarrow$$
  $\bigcirc$  Materials  $\rightarrow \stackrel{\frown}{\equiv}$  air  $\rightarrow$  Create/Edit...

Create/Edit Materials				
Name		Material Type		Order Materials by
air		fluid	•	Name
Chemical Formula		Fluent Fluid Materials		Chemical Formula
		air	•	Fluent Database
		Mixture		
		none	<b>*</b>	User-Defined Database
Properties				
Density (kg/m3)	onstant	▼ Edit		
1	.225			
Viscosity (kg/m-s)	constant	▼ Edit		
1	.7894e-05			
		Change/Create Delete Close	Help	.4

### Tip

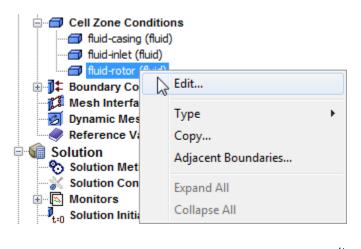
If needed, you could modify the fluid properties for air or copy another material from the database.

2. Click Close to close the Create/Edit Materials dialog box.

For details, see physical properties in the Fluent User's Guide.

# **10.4.6. Specifying Cell Zone Conditions**

1. Define the boundary conditions for the rotational reference frame (fluid-rotor).



Setup  $\rightarrow$  Cell Zone Conditions  $\rightarrow$  fluid-rotor  $\stackrel{\textcircled{}}{\hookrightarrow}$  Edit...

E Fluid			×
Zone Name			
fluid-rotor			
Material Name air	▼ Edit		
Frame Motion 📃 Lam	inar Zone 🔲 Source Terms		
Mesh Motion	Fixed Values		
Porous Zone			
Reference Frame Mi	esh Motion Porous Zone 3D	3D Fan Zone Embedded LES Reaction Source Terms Fixed Values Multipha	se
	constant	Translational Velocity X (m/s) 0 constant Y (m/s) 0 constant Y	
		OK Cancel Help	.4

#### a. Enable Frame Motion.

The dialog box will expand to show the relevant inputs.

b. Under the **Reference Frame** tab, retain the **Rotation-Axis Origin** default setting of (0,0).

This is the center of curvature for the circular boundaries of the rotating zone.

c. Enter -2500 rpm for Speed in the Rotational Velocity group box.

#### Note

The speed is entered as a negative value because the rotor is rotating clockwise, which is in the negative sense about the Z axis.

d. Click **OK** to close the **Fluid** dialog box.

#### Note

Since the other fluid zones are stationary, you do not need to set any boundary conditions for them. If one or more of the remaining fluid zones were also rotating, you would need to set the appropriate rotational speed for them.

#### Tip

In this example, the names of the fluid-zones in the mesh file leave no ambiguity as to which is the rotating fluid zone. In the event that you have a mesh without clear names, you may have difficulty identifying the various fluid-zones. Unlike interior zones, the fluid-zones cannot be individually selected and displayed from the **Mesh Display** dialog box. However, you can use commands in the text interface to display them.

### Setting Up Domain → Display

- i. Deselect all surfaces except, **default-interior**, **default-interior:013**, and **default-interior:015** in the **Surfaces** selection list to deselect them.
- ii. Click **Display**.

Only the domain boundaries and interface zones will be displayed.

- iii. Press Enter to get the > prompt.
- iv. Type the commands, in the console, as shown.

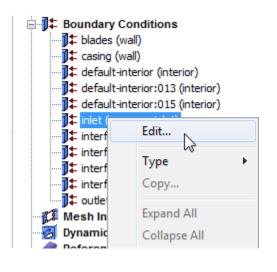
```
> display
/display> zone-mesh
()
zone id/name(1) [()] 4
zone id/name(2) [()] <Enter>
```

The resulting display shows that the zone with ID 4 (in this case **fluid-rotor**) corresponds to the rotating region.

v. Close the Mesh Display dialog box.

### **10.4.7. Setting Boundary Conditions**

1. Set the boundary conditions for the flow inlet (**inlet**) as specified in the problem description (see Figure 10.1: Schematic of the Problem (p. 430)).





Pressure Inlet	:							×
Zone Name						_		
inlet								
Momentum	Thermal	Radiation	Species	DPM	Multiph	lase	Potential	UDS
	Referer	nce Frame Ab	solute					•
Gau	uge Total Pre	ssure (pascal)	0			const	ant	•
Supersonic/Init	ial Gauge Pre	ssure (pascal)	0			const	ant	•
Directio	n Specificatio	n Method No	rmal to Bou	undary				•
	Turbulence							
	Specification	n Method Inte	ensity and \	iscosity l	Ratio			-
			Turbulent I	Intensity	(%) 5			P
		T	Turbulent \	/iscosity F	Ratio 10			P
		ОК	Cancel	Help				

- a. Review the boundary condition definition for the pressure-inlet type. Leave the settings at their defaults.
- b. Click **OK** to close the **Pressure Inlet** dialog box.

#### Note

All pressures that you specify in ANSYS Fluent are gauge pressures, relative to the operating pressure specified in the **Operating Conditions** dialog box. By default, the operating pressure is 101325 Pa.

For details, see operating pressure in the Fluent User's Guide.

2. Review and retain the default values for the boundary conditions for the flow outlet (**outlet**) so that the backflow turbulence parameters for the flow outlet (**outlet**) are set to the same values used for **inlet**.

#### Note

The backflow values are used only if reversed flow occurs at the outlet, but it is a good idea to use reasonable values, even if you do not expect any backflow to occur.

3. Define the velocity of the wall zone representing the blades (**blades**) relative to the moving fluid zone.

**Setup**  $\rightarrow$  Boundary Conditions  $\rightarrow$  blades  $\stackrel{\frown}{\rightarrow}$  Edit...

🛄 Wall	
Zone Name	
blades	
Adjacent Cell Zone	
fluid-rotor	
Momentum The	ermal Radiation Species DPM Multiphase UDS Wall Film Potential
Wall Motion	Motion
Stationary Wall	Relative to Adjacent Cell Zone Speed (rpm) 0     constant
Moving Wall	Absolute     Rotation-Axis Origin
	Translational X (m) 0
	Rotational Y (m) 0
	© Components
Shear Condition No Slp Specified Shear Specularity Coef Marangoni Stres	fficient
Wall Roughness	
Roughness Height (	(m) 0 constant 👻
Roughness Consta	ant 0.5 constant 🔻
	OK Cancel Help

With fluid-rotor set to a rotating reference frame, blades becomes a moving wall.

a. Select Moving Wall in the Wall Motion group box.

The **Wall** dialog box will expand to show the wall motion parameters.

- b. Retain the default selection of **Relative to Adjacent Cell Zone** and select **Rotational** in the **Motion** group box.
- c. Retain the default value of 0 rpm for (relative) Speed.
- d. Click **OK** to close the **Wall** dialog box.

The **Rotation-Axis Origin** should be located at x = 0 m and y = 0 m. With these settings, the blades will move at the same speed as the surrounding fluid.

# 10.4.8. Defining Mesh Interfaces

Recall that the fluid domain is defined as three distinct fluid zones. You must define mesh interfaces between the adjacent fluid zones so that ANSYS Fluent can solve the flow equations across the interfaces.

1. Set up the mesh interface between **fluid-inlet** and **fluid-rotor**.

**E** Setup  $\rightarrow$  Mesh Interfaces  $\stackrel{\red}{\rightarrow}$  New...

Mesh Interface	Interface Zones Side 1		Interface Zones Side 2	
int1	interface-1		interface-2	
	<b>x</b> [1/4]	= -, -,	[1/4]	= = =
	interface-1 interface-2 interface-3 interface-4		interface-1 interface-2 interface-3 interface-4	
Interface Options	Boundary Zones Side 1		Interface Wall Zones Side 1	
Periodic Boundary Condition Periodic Repeats	Boundary Zones Side 2		Interface Wall Zones Side 2	
Coupled Wall Matching Mapped Static			Interface Interior Zones	
Periodic Boundary Condition Type Offset Offset Translational X (m) 0 Rotational	Y (m) 0			
Auto Compute Offset				
Mapped  Enable Local Tolerance  Local Edge Length Factor				
	Create Delete Draw Lis	t Close He		

- a. Enter intl under **Mesh Interface** to name this interface definition.
- b. Select interface-1 for Interface Zone 1 and interface-2 for Interface Zone 2.

You can use the **Draw** button to help identify the interface-zones.

- c. Click **Create** in order to create the mesh interface, int1.
- d. In a similar manner, define a mesh interface called int2 between interface-3 and interface-4.
- e. Close the Create/Edit Mesh Interfaces dialog box.

### 10.4.9. Obtaining the Solution

1. Set the solution parameters.



Sol	lution	Met	hods
30	uuoi	meu	liuus

Pressure-Velocity Coupling
Scheme
Coupled
Spatial Discretization
Gradient
Least Squares Cell Based 🔹
Pressure
Second Order
Momentum
Second Order Upwind 🔹
Turbulent Kinetic Energy
Second Order Upwind 🔹
Turbulent Dissipation Rate
Second Order Upwind 🔹
Transient Formulation
Non-Iterative Time Advancement
Frozen Flux Formulation
Pseudo Transient
Warped-Face Gradient Correction
High Order Term Relaxation Options
Default
Help

- a. Select **Coupled** from the **Scheme** drop-down list in the **Pressure-Velocity Coupling** group box.
- b. Select Second Order Upwind for **Turbulent Kinetic Energy** and **Turbulent Dissipation Rate** in the **Spatial Discretization** group box.

The second-order scheme will provide a more accurate solution.

2. Enable that plotting of residuals during the calculation.

<b>Solving</b> $\rightarrow$ Reports $\rightarrow$ Residuals.
---

Residual Monitors					<b>-X</b>
Options Print to Console	Equations Residual	Monito	r Check Convergend	e Absolute Criteria	•
V Plot	continuity	V		5e-5	
Window	x-velocity	V		0.001	=
1 Curves Axes	y-velocity	V		0.001	
Iterations to Plot	k	V		0.001	
1000 🗢	epsilon	V		0.001	
Iterations to Store	Residual Values		_	onvergence Criterion bsolute	-
	Scale			Convergence Conditi	ons
	Compute Local	Scale			
OK Plot Renormalize Cancel Help					

- a. Ensure that **Plot** is enabled in the **Options** group box.
- b. Enter 5e-5 under Absolute Criteria for the continuity equation.

For this problem, the default value of 0.001 is insufficient for the flow rate in the blower to fully converge. All other settings should remain at their default values.

- c. Click **OK** to close the **Residual Monitors** dialog box.
- 3. Create a surface report definition and plot the volume flow rate at the flow outlet.

Solving  $\rightarrow$  Reports  $\rightarrow$  Definitions  $\rightarrow$  New  $\rightarrow$  Surface Report  $\rightarrow$  Volume Flow Rate

Surface Report Definition	
Name	Report Type
surf-mon-1	Volume Flow Rate
Options	Custom Vectors
	Vectors of
Per Surface	· · · · · · · · · · · · · · · · · · ·
Average Over	Custom Vectors
1	Field Variable
Report Files [0/0]	Pressure
	Static Pressure 👻
	Surfaces Filter Text
	interface-2
	interface-3
Report Plots [0/0]	interface-4
	default-interior
	default-interior:013
	default-interior:015
	int1-interior-1-1
Create	int2-interior-1-1
Report File	Outlet     outlet
Report Plot	▲ Wall
Frequency 1	blades
Print to Console	casing 👻
Create Output Parameter	New Surface 🔻
OK	Compute Cancel Help

- a. Enter **surf-mon-1** for the **Name** of the surface report definition.
- b. In the Create group box, enable Report File, Report Plot and Print to Console.

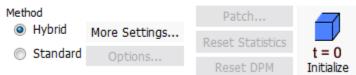
When the **Report File** option is selected in the **Surface Report Definition** dialog box, the volume flow rate history will be written to a file. If you do not enable the **Report File** option, the history information will be lost when you exit ANSYS Fluent.

- d. Select outlet from the Surfaces selection list.
- e. Click **OK** to save the surface report definition settings and close the **Surface Report Definition** dialog box.
- 4. Initialize the solution.

Solving → Initialization

c. Note

Initialization



- a. Retain the **Method** at the default of **Hybrid** in the **Initialization Methods** group.
- b. Click Initialize to initialize the solution.

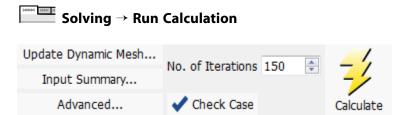
### Note

For flows in complex topologies, hybrid initialization will provide better initial velocity and pressure fields than standard initialization. This in general will help in improving the convergence behavior of the solver.

5. Save the case file (blower.cas.gz).



6. Start the calculation by requesting 150 iterations.



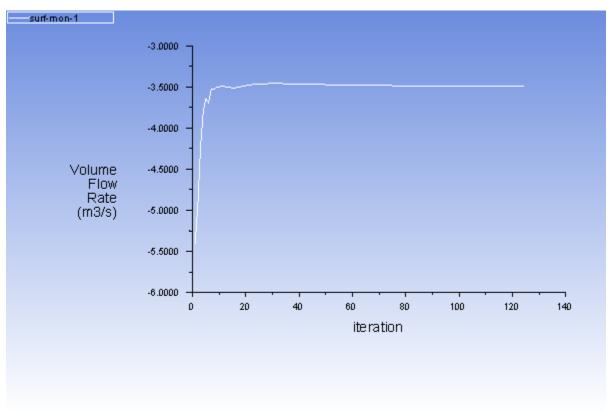
a. Enter 150 for **No. of Iterations**.

#### b. Click Calculate.

Early in the calculation, ANSYS Fluent will report that there is reversed flow occurring at the exit. This is due to the sudden expansion, which results in a recirculating flow near the exit.

The solution will converge in approximately 120 iterations (when all residuals have dropped below their respective criteria).





The surface report history indicates that the flow rate at the outlet has ceased changing significantly, further indicating that the solution has converged. The volume flow rate is approximately  $3.49 \text{ m}^3$ /s.

### Note

You can examine the residuals history in an appropriate graphics tab window.

7. Save the case and data files (blower2.cas.gz and blower2.dat.gz).

File 
$$\rightarrow$$
 Write  $\rightarrow$  Case & Data...

#### Note

It is good practice to save the case file whenever you are saving the data. This will ensure that the relevant parameters corresponding to the current solution data are saved accordingly.

# 10.4.10. Step 9: Postprocessing

1. Display filled contours of static pressure (Figure 10.4: Contours of Static Pressure (p. 448)).

```
Postprocessing \rightarrow Graphics \rightarrow Contours \rightarrow Edit...
```

Contours					×
Options          Image: Options <t< th=""><th>Contours of Pressure</th><th></th><th></th><th></th><th>•</th></t<>	Contours of Pressure				•
	Static Pressure				•
Auto Range     Clip to Range	Min (pascal) -1923.915	Max (pascal) 197.9079			
Draw Profiles	Surfaces Filter Text		-0	F. 🐬	<b>-x</b>
Coloring Banded Smooth Levels Setup 20 $\textcircled{\Rightarrow}$ 1 $\textcircled{\Rightarrow}$	<ul> <li>Inlet inlet</li> <li>Interface interface-1 interface-2 interface-3 interface-4</li> <li>New Surface      </li> <li>Display Compute</li> </ul>	Close Help	2		• E

- a. Enable **Filled** in the **Options** group box.
- b. Ensure **Pressure...** and **Static Pressure** are selected from the **Contours of** drop-down lists.
- c. Click **Display** and close the **Contours** dialog box (see Figure 10.4: Contours of Static Pressure (p. 448)).

Pressure distribution in the flow domain is plotted in graphics window.

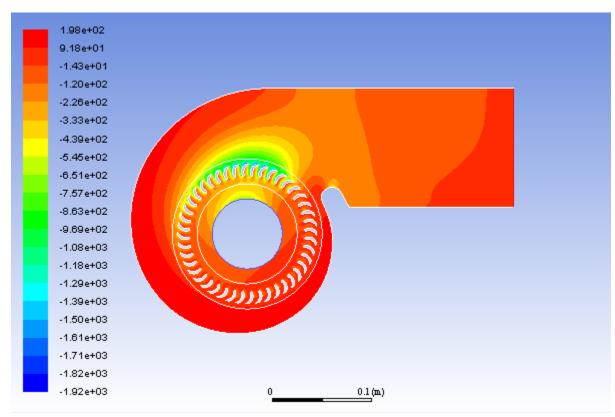


Figure 10.4: Contours of Static Pressure

2. Display absolute velocity vectors (Figure 10.5: Velocity Vectors (p. 450)).

Postprocessing  $\rightarrow$  Graphics  $\rightarrow$  Vectors  $\rightarrow$  Edit...

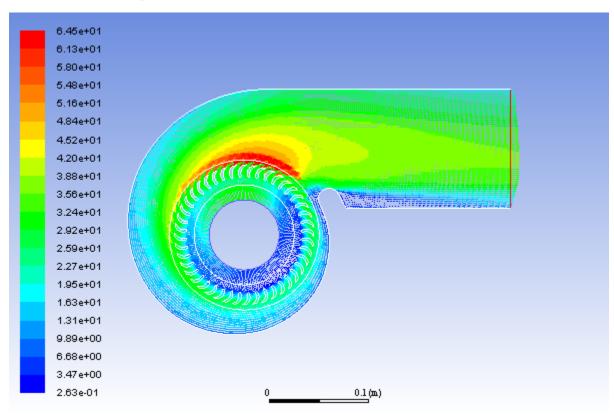
<b>Q</b> Vectors		<b>—X</b> —
Options Global Range Auto Range Clip to Range Auto Scale Draw Mesh	Vectors of Velocity	
	Color by Velocity	•
	Velocity Magnitude	•
	Min (m/s)	Max (m/s)
Style	0.2629554	64.46782
arrow 🔻 Scale Skip	Surfaces Filter Text	
10 0 🜩 Vector Options	<ul> <li>✓ Inlet inlet</li> <li>✓ Interface</li> </ul>	
Custom Vectors	interface-1	
	interface-3	-
	New Surface 🔻	
Display Compute Close Help		

a. Enter 10 for Scale.

By default, **Auto Scale** is chosen. This will automatically scale the length of velocity vectors relative to the size of the smallest cell in the mesh. To increase the length of the "scaled" vectors, set the **Scale** factor to a value greater than 1.

- b. Retain the default selection of **Velocity** from the **Vectors of** drop-down list.
- c. Retain the default selection of Velocity... and Velocity Magnitude from the Color by drop-down list.
- d. Click **Display** and close the **Vectors** dialog box (see Figure 10.5: Velocity Vectors (p. 450)).

### **Figure 10.5: Velocity Vectors**



The velocity vectors show an area of flow separation near the bottom of the outlet duct. You can zoom in on this area and see the flow recirculation.

- 3. Display relative velocity vectors with respect to the rotational reference frame (**fluid-rotor**).
  - a. In the Reference Values task page, select fluid-rotor from the Reference Zone drop-down list.

b. Open the **Vectors** dialog box.

Postprocessing  $\rightarrow$  Graphics  $\rightarrow$  Vectors  $\rightarrow$  Edit...

Vectors		<b>-X</b> -			
Options Global Range Auto Range Clip to Range	Vectors of Relative Velocity Color by Velocity				
Auto Scale Draw Mesh	Relative Velocity Magnitude	•			
Style	Min (m/s) Max (m 0.1527302 72.051				
Scale Skip	Surfaces Filter Text				
Vector Options	<ul> <li>Inlet</li> <li>inlet</li> <li>Interface</li> </ul>				
Custom Vectors	interface-1 interface-2 interface-3	+			
	New Surface 🔻				
	Display Compute Close	Help			

- i. Select Relative Velocity from the Vectors of drop-down list.
- ii. Select Velocity... and Relative Velocity Magnitude from the Color by drop-down list.
- iii. Set Scale to 2.
- iv. Click **Display** and close the **Vectors** dialog box.

The relative air velocity vectors viewed in the frame of reference rotating with the rotor are displayed.

v. Zoom in on the rotor blade region as shown in Figure 10.6: Relative Velocity Vectors (p. 452) and examine the air flow through the rotor blade passages.

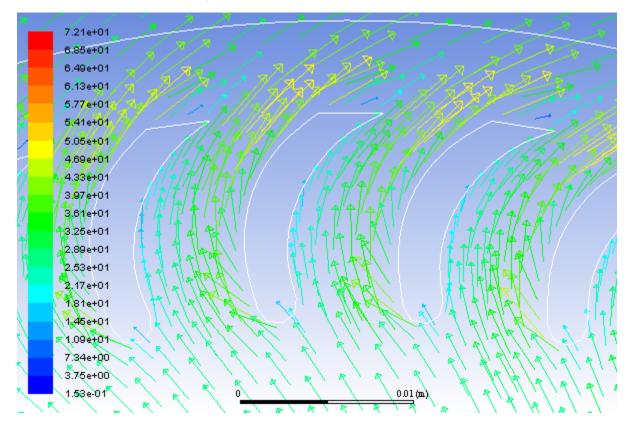
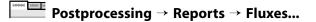


Figure 10.6: Relative Velocity Vectors

4. Report the mass flux at **inlet** and **outlet**.



Flux Reports				×
Options     Mass Flow Rate	Boundaries Filter Text	- <b>x</b>	Results	
Total Heat Transfer Rate	default-interior:015	*		•
Radiation Heat Transfer Rate	inlet		4.276740550994873	
	int1-interior-1-1			
	int1-side1-wall-interface-1			
	int1-side2-wall-interface-2 int2-interior-1-1			
	int2-interior-1-1 int2-side1-wall-interface-3			
	interface-1			
	interface-2	E		E
	interface-3			
	interface-4	_		
	outlet			
	wall-21	-	-4.276749134063721	-
	•	P	٠	•
Save Output Parameter			Net Results (kg/s)	
			-8.583069e-06	
	Compute Write Close Help			

- a. Retain the selection of Mass Flow Rate in the Options group box.
- b. Select inlet and outlet in the Boundaries selection list.
- c. Click Compute.

The net mass imbalance should be no more than a small fraction (say, 0.5%) of the total flux through the system. If a significant imbalance occurs, you should decrease your residual tolerances by at least an order of magnitude and continue iterating.

The flux report will compute fluxes only for boundary zones.

d. Close the **Flux Reports** dialog box.

#### Note

You can use the **Surface Integrals** option to report fluxes on surfaces or planes.

Results  $\rightarrow$  Reports  $\rightarrow$  Surface Integrals  $\stackrel{\frown}{\rightarrow}$  Edit...

### 10.5. Summary

This tutorial illustrates the procedure for setting up and solving problems with multiple reference frames using ANSYS Fluent. Although this tutorial considers only one rotating fluid zone, extension to multiple rotating fluid zones is straightforward as long as you delineate each fluid zone.

Note that this tutorial was solved using the default absolute velocity formulation. For some problems involving rotating reference frames, you may want to use the relative velocity formulation. See the Fluent User's Guide for details.

## **10.6. Further Improvements**

This tutorial guides you through the steps to reach an initial solution. You may be able to obtain a more accurate solution by using an appropriate higher-order discretization scheme and by adapting the mesh. Mesh adaption can also ensure that the solution is independent of the mesh. These steps are demonstrated in Introduction to Using ANSYS Fluent: Fluid Flow and Heat Transfer in a Mixing Elbow (p. 121).

# **Chapter 11: Using Sliding Meshes**

This tutorial is divided into the following sections:

- 11.1. Introduction
- 11.2. Prerequisites
- 11.3. Problem Description
- 11.4. Setup and Solution
- 11.5. Summary
- 11.6. Further Improvements

# 11.1. Introduction

The analysis of turbomachinery often involves the examination of the transient effects due to flow interaction between the stationary components and the rotating blades. In this tutorial, the sliding mesh capability of ANSYS Fluent is used to analyze the transient flow in an axial compressor stage. The rotorstator interaction is modeled by allowing the mesh associated with the rotor blade row to rotate relative to the stationary mesh associated with the stator blade row.

This tutorial demonstrates how to do the following:

- Create periodic zones.
- Set up the transient solver and cell zone and boundary conditions for a sliding mesh simulation.
- Set up the mesh interfaces for a periodic sliding mesh model.
- Sample the time-dependent data and view the mean value.

# 11.2. Prerequisites

This tutorial is written with the assumption that you have completed one or more of the introductory tutorials found in this manual:

- Introduction to Using ANSYS Fluent in ANSYS Workbench: Fluid Flow and Heat Transfer in a Mixing Elbow (p. 1)
- Parametric Analysis in ANSYS Workbench Using ANSYS Fluent (p. 73)
- Introduction to Using ANSYS Fluent: Fluid Flow and Heat Transfer in a Mixing Elbow (p. 121)

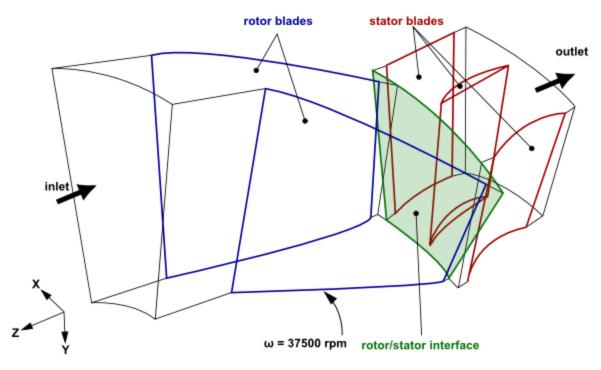
and that you are familiar with the ANSYS Fluent tree and ribbon structure. Some steps in the setup and solution procedure will not be shown explicitly.

# 11.3. Problem Description

The model represents a single-stage axial compressor composed of two blade rows. The first row is the rotor with 16 blades, which is operating at a rotational speed of 37,500 rpm. The second row is the stator with 32 blades. The blade counts are such that the domain is rotationally periodic, with a periodic

angle of 22.5 degrees. This enables you to model only a portion of the geometry, namely, one rotor blade and two stator blades. Due to the high Reynolds number of the flow and the relative coarseness of the mesh (both blade rows are composed of only 13,856 cells total), the analysis will employ the inviscid model, so that ANSYS Fluent is solving the Euler equations.





## 11.4. Setup and Solution

The following sections describe the setup and solution steps for this tutorial:

11.4.1. Preparation
11.4.2. Mesh
11.4.3. General Settings
11.4.4. Models
11.4.5. Materials
11.4.6. Cell Zone Conditions
11.4.7. Boundary Conditions
11.4.8. Operating Conditions
11.4.9. Mesh Interfaces
11.4.10. Solution
11.4.11. Postprocessing

### 11.4.1. Preparation

To prepare for running this tutorial:

1. Set up a working folder on the computer you will be using.

2. Go to the ANSYS Customer Portal, https://support.ansys.com/training.

#### Note

If you do not have a login, you can request one by clicking **Customer Registration** on the log in page.

- 3. Enter the name of this tutorial into the search bar.
- 4. Narrow the results by using the filter on the left side of the page.
  - a. Click ANSYS Fluent under Product.
  - b. Click **18.0** under **Version**.
- 5. Select this tutorial from the list.
- 6. Click the sliding\_mesh\_R180.zip link to download the input files.
- 7. Unzip sliding\_mesh\_R180.zip to your working folder.

The mesh file axial\_comp.msh can be found in the sliding\_mesh directory created after unzipping the file.

8. Use Fluent Launcher to start the **3D** version of ANSYS Fluent.

Fluent Launcher displays your **Display Options** preferences from the previous session.

For more information about Fluent Launcher, see starting ANSYS Fluent using the Fluent Launcher in the Fluent Getting Started Guide.

- 9. Ensure that the **Display Mesh After Reading** and **Workbench Color Scheme** options are enabled.
- 10. Enable single precision (disable **Double Precision**).
- 11. Do not enable **Meshing Mode**.
- 12. Run in Serial under Processing Options.

### 11.4.2. Mesh

1. Read in the mesh file axial\_comp.msh.

File  $\rightarrow$  Read  $\rightarrow$  Mesh...

### 11.4.3. General Settings

1. Check the mesh.

Setting Up Domain  $\rightarrow$  Mesh  $\rightarrow$  Check

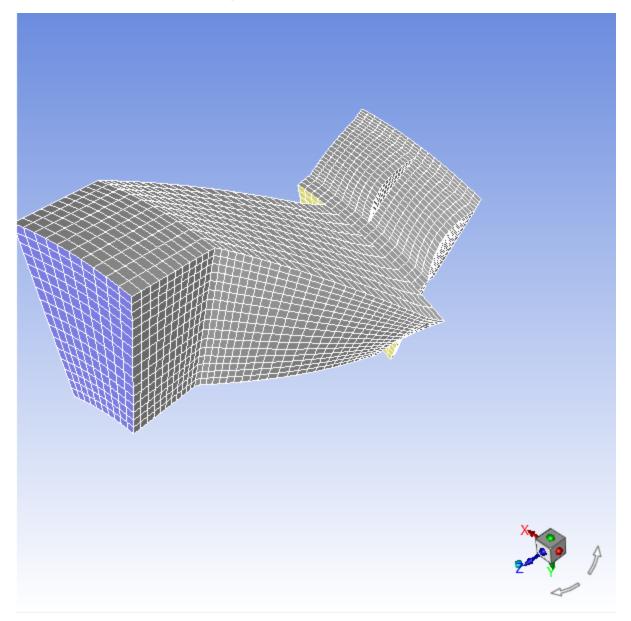
ANSYS Fluent will perform various checks on the mesh and will report the progress in the console. Ensure that the reported minimum volume is a positive number.

Warnings will be displayed regarding unassigned interface zones, resulting in the failure of the mesh check. You do not need to take any action at this point, as this issue will be rectified when you define the mesh interfaces in a later step.

2. Examine the mesh (Figure 11.2: Rotor-Stator Display (p. 458)).

Orient the view to display the mesh as shown in Figure 11.2: Rotor-Stator Display (p. 458). The inlet of the rotor mesh is colored blue, the interface between the rotor and stator meshes is colored yellow, and the outlet of the stator mesh is colored red.

Figure 11.2: Rotor-Stator Display



3. Use the text user interface to change zones rotor-per-1 and rotor-per-3 from wall zones to periodic zones.

- a. Press **Enter** in the console to get the command prompt (>).
- b. Type the commands as shown below in the console:

> mesh/modify-zones/list-	-zones
---------------------------	--------

id	name	type	material	kin
13	fluid-rotor	fluid	air	cel
28	fluid-stator	fluid	air	cel
2	default-interior:0	interior		fac
15	default-interior	interior		fac
3	rotor-hub	wall	air	fac
4	rotor-shroud	wall	air	fac
7	rotor-blade-1	wall	air	fac
8	rotor-blade-2	wall	air	fac
16	stator-hub	wall	air	fac
17	stator-shroud	wall	air	fac
20	stator-blade-1	wall	air	fac
21	stator-blade-2	wall	air	fac
22	stator-blade-3	wall	air	fac
23	stator-blade-4	wall	air	fac
5	rotor-inlet	pressure-inlet		fac
19	stator-outlet	pressure-outle	t	fac
10	rotor-per-1	wall	air	fac
12	rotor-per-2	wall	air	fac
24	stator-per-2	wall	air	fac
26	stator-per-1	wall	air	fac
б	rotor-interface	interface		fac
18	stator-interface	interface		fac
11	rotor-per-4	wall	air	fac
9	rotor-per-3	wall	air	fac
25	stator-per-4	wall	air	fac
27	stator-per-3	wall	air	fac

Rotational periodic? (if no, translational) [yes] **yes** Create periodic zones? [yes] **yes** 

zone 9 deleted

created periodic zones.

4. Similarly, change the following wall zone pairs to periodic zones:

Zone Pairs	Respective Zone IDs
rotor-per-2 and rotor-per-4	12 and 11
stator-per-1 and stator-per-3	26 and 27
stator-per-2 and stator-per-4	24 and 25

5. Define the solver settings.

General		
Mesh		
Scale	Check	Report Quality
Display		
Solver		
Туре	Velo	ocity Formulation
Pressure-Base	sed 💿	Absolute
Density-Base	ed 💿	Relative
Time		
Steady		
Transient		
Gravity Units	5	
Help		

- a. Retain the default selection of **Pressure-Based** in the **Type** list.
- b. Select Transient in the Time list.
- 6. Define the units for the model.

Setup →	<b>\$</b> General →	Units
---------	---------------------	-------

Set Units			×
Quantities acceleration angle angular-velocity area area-inverse collision-rate concentration contact-resistance crank-angle crank-angular-velocity		Units rad/s deg/s rpm Factor 0.1047198	Set All to default si british cgs
density density-gradient	New	Close Help	

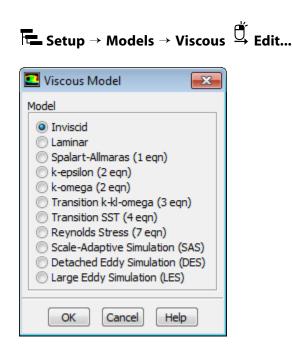
- a. Select **angular-velocity** from the **Quantities** selection list.
- b. Select **rpm** from the **Units** selection list.
- c. Select **pressure** from the **Quantities** selection list.

Scroll down the **Quantities** list to find **pressure**.

- d. Select atm from the Units selection list.
- e. Close the Set Units dialog box.

### 11.4.4. Models

1. Enable the inviscid model.



- a. Select Inviscid in the Model list.
- b. Click **OK** to close the **Viscous Model** dialog box.

### 11.4.5. Materials

1. Specify air (the default material) as the fluid material, using the ideal gas law to compute density.

```
E Setup \rightarrow \bigcirc Materials \rightarrow Fluids \rightarrow \stackrel{\frown}{=} air \rightarrow \stackrel{\frown}{\subseteq} Edit...
```

Create/Edit Materials		<b>X</b>
Name	Material Type	Order Materials by
Chemical Formula	fluid FLUENT Fluid Materials	Chemical Formula
	air	
	Mixture	User-Defined Database
	none	*
Properties		
Density (kg/m3)	ideal-gas	
Cp (Specific Heat) (j/kg-k)	constant	
Molecular Weight (kg/kgmol)	1006.43	
	28.966	
	Change/Create Delete Close	Help

- a. Retain the default entry of air in the Name text entry field.
- b. Select **ideal-gas** from the **Density** drop-down list in the **Properties** group box.
- c. Retain the default values for all other properties.
- d. Click Change/Create and close the Create/Edit Materials dialog box.

As reported in the console, ANSYS Fluent will automatically enable the energy equation, since this is required when using the ideal gas law to compute the density of the fluid.

### 11.4.6. Cell Zone Conditions

Setup  $\rightarrow \textcircled{+}$  Cell Zone Conditions

- • • • • • - • - •
Cell Zone Conditions
Zone Filter Text
fluid-rotor
fluid-stator
Phase Type ID
mixture V fluid V 13
Edit Copy Profiles
Parameters
Operating Conditions
Display Mesh
Porous Formulation
Superficial Velocity
O Physical Velocity
Help

1. Set the cell zone conditions for the fluid in the rotor (**fluid-rotor**).

Setup →	↓ Cell Zone	Conditions	→ 🔚 fluid	-rotor →	Edit
Jetup /	• Cell Zolle	conditions			Luit

E Fluid										<b>X</b>
Zone Name										
fluid-rotor										
Material Name air		<ul> <li>Edit</li> </ul>	· )							
Frame Motion	3D Fan Zone 📃	Source	Terms							
Mesh Motion		Fixed Va	alues							
Porous Zone										
Reference Frame	Mesh Motion	Porous	Zone	3D Fan Zon	e Embedo	ded LES	Reaction	Source Terms	Fixed Values	Multiphase
Relative Specificat	ion		UDF							
Relative To Cell Zor				Motion Functio	none		1			
							J			
Rotation-Axis Origi				Rotation-Axis D						
X (m) 0	constant	-		X 0	constant	•				
Y (m) 0	constant	-		YO	constant	•				
Z (m) 0	constant	•	2	Z 1	constant	•				
Rotational Velocity	/			Translational	Velocity					
Speed (rpm) 3750	0 consta	nt	•	X (m/s) 0	CO	nstant	•			
Copy To Frame Mo	tion			Y (m/s) 0	co	nstant	•			
				Z (m/s) 0	CO	nstant	•			
				_						
				OK	Cancel	Help				

- a. Enable Mesh Motion.
- b. Click the **Mesh Motion** tab.
- c. Retain the default values of (0, 0, 1) for X, Y, and Z in the Rotation-Axis Direction group box.
- d. Enter 37500 rpm for **Speed** in the **Rotational Velocity** group box.
- e. Click **OK** to close the **Fluid** dialog box.
- 2. Set the cell zone conditions for the fluid in the stator (fluid-stator).

**F**Setup  $\rightarrow$  **Cell Zone Conditions**  $\rightarrow \stackrel{\frown}{=}$  fluid-stator  $\rightarrow$  Edit...

E Fluid Zone Name fluid-stator Material Name air	▼) Edit	•••		)				<b></b>
Frame Motion 3D								
Mesh Motion Porous Zone	Fixed Va	lues						
	lesh Motion Porous	7000	3D Fan Zone	Embedded LES	Reaction	Source Terms	Fixed Values	Multiphase
	resit Modori   Porous	zone	SU Part Zorre	Embedded LES	Reaction	Source terms	Poceu values	Multipliase
Rotation-Axis Origin			n-Axis Direction					
X (m) 0	constant 💌	X 0	const	ant 🔹				
Y (m) 0	constant 🔹	Y 0	const	ant 🔹				
Z (m) 0	constant 🔻	Z 1	const	ant 👻				
			OK	Cancel Help				

- a. Retain the default values of (0, 0, 1) for X, Y, and Z in the Rotation-Axis Direction group box.
- b. Click **OK** to close the **Fluid** dialog box.

### **11.4.7. Boundary Conditions**

Setup → < Boundary Conditions

Boundary Conditions	
Zone Filter Text	; =
default-interior	
default-interior:0	
rotor-blade-1	
rotor-blade-2	
rotor-hub	
rotor-inlet	
rotor-interface	=
rotor-per-1	
rotor-per-2	
rotor-shroud	
stator-blade-1	
stator-blade-2 stator-blade-3	
stator-blade-5	
stator-hub	
stator-interface	
	*
Phase Type ID	
mixture 👻 🔽 –1	
Edit Copy Profiles	
Parameters Operating Conditions	
Display Mesh Periodic Conditions	
Highlight Zone	
Help	

1. Enter rotor-inlet into the **Zone** field to filter the zone list.



Pressure Inlet							×
Zone Name							
rotor-inlet							
Momentum Thermal Ra	diation	Species	DPM	Multi	phase	Potential	UDS
Reference Fra	me Abso	lute					-
Gauge Total Pressur	e (atm)	1			constan	it	-
Supersonic/Initial Gauge Pressur	Supersonic/Initial Gauge Pressure (atm) 0.9				constant 🔹		
Direction Specification Meth	od Norm	nal to Bound	ary				-
Acoustic Wave Model							
Off							
Non Reflecting							
<ul> <li>Impedance</li> <li>Transparent Flow Forcing</li> </ul>							
L	ОК	Cancel	Help				

- a. Enter 1.0 atm for **Gauge Total Pressure**.
- b. Enter 0.9 atm for Supersonic/Initial Gauge Pressure.

For information about the Supersonic/Initial Gauge Pressure, see the Fluent User's Guide.

c. Click the **Thermal** tab and enter 288 K for **Total Temperature**.

Pressure Inle	t						×
Zone Name							
rotor-inlet							
Momentum	Thermal	Radiation	Species	DPM	Multiphase	Potential	UDS
Total Tempera	ture (k) 288			constant		•	
		OK	Cancel	Help			

- d. Click **OK** to close the **Pressure Inlet** dialog box.
- 2. Enter stator-outlet into the **Zone** field to filter the zone list.

Setup →	Boundary Conditions	$\rightarrow \equiv$ stator-outlet $\rightarrow$ Edit
---------	---------------------	---

Pressure Out	let						×
Zone Name							
stator-outlet							
Momentum	Thermal	Radiation	Species	DPM	Multiphase	Potential	UDS
В	ackflow Refe	rence Frame	Absolute				•
	Gaug	e Pressure (a	tm) 1.08		CO	onstant	•
Backflow Direct	tion Specifica	tion Method	Normal to B	loundary			•
🔽 Radial Equili	brium Pressur	e Distribution	I.				
Average Pre	essure Specifi	ication					
Target Mass	Flow Rate						
Acoustic Way	ve Model						
Off							
Non Refle	-						
<ul> <li>Impedance</li> <li>Transpare</li> </ul>	e nt Flow Forci						
Tanspare		'Y					
		C	Cance	l Help	]		

- a. Enter 1.08 atm for **Gauge Pressure**.
- b. Enable Radial Equilibrium Pressure Distribution.
- c. Click the Thermal tab and enter 288 K for Backflow Total Temperature.

Pressure Out	let						×
Zone Name							
stator-outlet							
Momentum	Thermal	Radiation	Species	DPM	Multiphase	Potential	UDS
Backflow Total	Temperatur	e (k) 288			constant	•	
		0	K Cance	Help	]		

d. Click **OK** to close the **Pressure Outlet** dialog box.

#### Note

The momentum settings and temperature you input at the pressure outlet will be used only if flow enters the domain through this boundary. It is important to set reasonable values for these downstream scalar values, in case flow reversal occurs at some point during the calculation.

3. Retain the default boundary conditions for all wall zones.

**Setup**  $\rightarrow$  **Conditions**  $\rightarrow$  **Edit...** 

💶 Wall								<b>×</b>
Zone Name								
rotor-blade-1								
Adjacent Cell Zo	one							
fluid-rotor								
Momentum	Thermal	Radiation	Species	DPM	Multiphase	UDS	Wall Film	Potential
			ок С	ancel	Help			

#### Note

For wall zones, ANSYS Fluent always imposes zero velocity for the normal velocity component, which is required whether or not the fluid zone is moving. This condition is all that is required for an inviscid flow, as the tangential velocity is computed as part of the solution.

## 11.4.8. Operating Conditions

1. Set the operating pressure.



Operating Conditions	<b>—</b> ×
Pressure	Gravity
Floating Operating Pressure Operating Pressure (atm)	Gravity
Reference Pressure Location	
X (m) 0	
Y (m) 0	
Z (m) 0 P	
OK Cancel Help	>

- a. Enter 0 atm for **Operating Pressure**.
- b. Click OK to close the Operating Conditions dialog box.

Since you have specified the boundary condition inputs for pressure in terms of absolute pressures, you have to set the operating pressure to zero. Boundary condition inputs for pressure should always be relative to the value used for operating pressure.

### 11.4.9. Mesh Interfaces

1. Create a periodic mesh interface between the rotor and stator mesh regions.

Create/Edit Mesh Inte	erfaces		
lesh Interface		Interface Zones Side 1	Interface Zones Side 2
int		rotor-interface	stator-interface
	- <u>x</u> -	[1/2]	
		rotor-interface	rotor-interface
		stator-interface	stator-interface
Interface Options Periodic Boundary C	andition.	Boundary Zones Side 1	Interface Wall Zones Side 1
Periodic Boundary C Periodic Repeats	ondition	Boundary Zones Side 2	Interface Wall Zones Side 2
Coupled Wall			
Matching			Interface Interior Zones
Mapped			
Static			
Periodic Boundary Con	ndition		
Туре	Offset		
<ul> <li>Translational</li> <li>Rotational</li> </ul>	X (m) 0	Y (m) 0	Z (m) 0
Auto Compute Offs	et		
Mapped			
Enable Local Tolerar	nce		
1			

- a. Enter int for **Mesh Interface**.
- b. Enable Periodic Repeats in the Interface Options group box.

Enabling this option, allows ANSYS Fluent to treat the interface between the sliding and non-sliding zones as periodic where the two zones do not overlap.

c. Select rotor-interface from the Interface Zones Side 1 selection list.

#### Note

In general, when one interface zone is smaller than the other, it is recommended that you choose the smaller zone as **Interface Zone 1**. In this case, since both zones are approximately the same size, the order is not significant.

d. Select **stator-interface** from the **Interface Zones Side 2** selection list.

- e. Click Create and close the Create/Edit Mesh Interfaces dialog box.
- 2. Check the mesh again to verify that the warnings displayed earlier have been resolved.

**Setting Up Domain**  $\rightarrow$  Mesh  $\rightarrow$  Check

### 11.4.10. Solution

1. Set the solution parameters.

**Solving**  $\rightarrow$  Solution  $\rightarrow$  Methods...

Solution Methods
Pressure-Velocity Coupling
Scheme
Coupled
Spatial Discretization
Gradient
Least Squares Cell Based 🔻
Pressure
Second Order
Density
Second Order Upwind 🔻
Momentum
Second Order Upwind
Energy
Second Order Upwind 🔹
Transient Formulation
First Order Implicit
Non-Iterative Time Advancement
Frozen Flux Formulation
Warped-Face Gradient Correction
High Order Term Relaxation Options
Default
Help

a. Select **Coupled** from the **Pressure-Velocity Coupling** group box.

For many general fluid-flow problems, convergence speed can be improved by using the coupled solver.

2. Change the Solution Controls

**Solving**  $\rightarrow$  Controls  $\rightarrow$  Controls...

Solution Controls
Flow Courant Number
200
Explicit Relaxation Factors
Momentum 0.5
Pressure 0.5
Under-Relaxation Factors
Density
1
Body Forces
1
Energy
1
Temperature
0.9
*
Default
Equations Limits Advanced
Help

- a. Enter 0.5 for **Momentum** and **Pressure** in the **Explicit Relaxation Factors** group box.
- b. Enter 0.9 for Temperature in the Under-Relaxation Factors group box.
- 3. Enable the plotting of residuals during the calculation.



Residual Monitors					×
Options	Equations				
Print to Console	Residual	Monitor	Check Convergence	e Relative Criteria	<u> </u>
V Plot	continuity	<b>V</b>	$\checkmark$	0.01	
Window	x-velocity			0.01	
1 Curves Axes	y-velocity	<b>V</b>		0.01	
Iterations to Plot	z-velocity	<b>V</b>		0.01	
1000 ≑		[ma]			
	Residual Values			Convergence Criterio	on
	Normalize		Iterations	relative	•
Iterations to Store			5		
1000 🚖	Scale			Convergence Cond	itions
Compute Local Scale					
OK Plot Renormalize Cancel Help					

- a. Ensure that the **Plot** is selected in the **Options** group box.
- b. Select relative from the Convergence Criterion drop-down list.
- c. Enter 0.01 for **Relative Criteria** for each **Residual** (continuity, x-velocity, y-velocity, z-velocity, and energy).
- d. Click **OK** to close the **Residual Monitors** dialog box.
- 4. Enable the plotting of mass flow rate at the inlet (rotor-inlet).

**Solving**  $\rightarrow$  Reports  $\rightarrow$  Definitions  $\rightarrow$  New  $\rightarrow$  Surface Report  $\rightarrow$  Mass Flow Rate...

Surface Report Definition		
Name	Report Type	
surf-mon-1	Mass Flow Rate	
Options	Custom Vectors	
	Vectors of	
Per Surface	<b></b>	
Average Over	Custom Vectors	
1	Field Variable	
Report Files [0/0]	Pressure	
	Static Pressure 👻	
	Surfaces rotor-inlet 🗙 🙃 🗮 📆	
	rotor-inlet	
Report Plots [0/0]		
Create		
✓ Report File		
Report Plot		
Frequency 1		
Print to Console	Highlight Surfaces	
Create Output Parameter	New Surface 🔻	
OK Compute Cancel Help		

- a. Enter **surf-mon-1** for the **Name** of the surface report definition.
- b. In the Create group box, enable Report File, Report Plot and Print to Console.
- c. Enter rotor-inlet in the Surfaces field to filter the list.
- d. Select rotor-inlet from the Surfaces selection list.
- e. Click **OK** to save the surface report definition settings and close the **Surface Report Definition** dialog box.

**surf-mon-1-rplot** and **surf-mon-1-rfile** that are automatically generated by Fluent appear in the tree (under **Solution/Monitors/Report Plots** and **Solution/Monitors/Report Files**, respectively).

5. Enable the plotting of mass flow rate at the outlet (stator-outlet).

Solving  $\rightarrow$  Reports  $\rightarrow$  Definitions  $\rightarrow$  New  $\rightarrow$  Surface Report  $\rightarrow$  Mass Flow Rate...

Surface Report Definition		
Name	Report Type	
surf-mon-2	Mass Flow Rate	
Options Per Surface Average Over 1 Report Files [0/1] Surf-mon-1-rfile	Custom Vectors Vectors of Custom Vectors Field Variable Pressure Static Pressure Surfaces stator-outlet	
Report Plots [0/1]	stator-outlet	
Create Create Report File Create Create Preport Plot Frequency Print to Console	Lighlight Surfaces	
	Highlight Surfaces	
Create Output Parameter	New Surface 🔻	
OK Compute Cancel Help		

- a. Enter **surf-mon-2** for the **Name** of the surface report definition.
- b. In the Create group box, enable Report File, Report Plot and Print to Console.
- c. Enter stator-outlet in the Surfaces field to filter the list.
- d. Select stator-outlet from the Surfaces selection list.
- e. Click **OK** to save the surface report definition settings and close the **Surface Report Definition** dialog box.

**surf-mon-2-rplot** and **surf-mon-2-rfile** that are automatically generated by Fluent appear in the tree (under **Solution/Monitors/Report Plots** and **Solution/Monitors/Report Files**, respectively).

6. Enable the plotting of the area-weighted average of the static pressure at the interface (**stator-interface**).

Solving  $\rightarrow$  Reports  $\rightarrow$  Definitions  $\rightarrow$  New  $\rightarrow$  Surface Report  $\rightarrow$  Area-Weighted Average...

Surface Report Definition		
Name	Report Type	
surf-mon-3	Area-Weighted Average	
Options	Custom Vectors	
	Vectors of	
Per Surface	<b></b>	
Average Over	Custom Vectors	
1		
	Field Variable	
Report Files [0/2]	Pressure▼	
surf-mon-1-rfile	Static Pressure 👻	
surf-mon-2-rfile	Surfaces stator-interface 🗙 💼 🗮 🐺	
	stator-interface	
Report Plots [0/0]		
Create		
Report File		
Report Plot		
Frequency 1 🚔		
Print to Console	Highlight Surfaces	
Create Output Parameter	New Surface 🔻	
OK Compute Cancel Help		

- a. Enter **surf-mon-3** for the **Name** of the surface report definition.
- b. In the Create group box, enable Report File, Report Plot and Print to Console.
- c. Retain the default selection of **Pressure...** and **Static Pressure** from the **Field Variable** drop-down lists.
- d. Enter stator-interface in the Surfaces field to filter the list.
- e. Select stator-interface from the Surfaces selection list.
- f. Click **OK** to save the surface report definition settings and close the **Surface Report Definition** dialog box.

**surf-mon-3-rplot** and **surf-mon-3-rfile** that are automatically generated by Fluent appear in the tree (under **Solution/Monitors/Report Plots** and **Solution/Monitors/Report Files**, respectively).

7. Initialize the solution using the values at the inlet (rotor-inlet).

Solution Initialization Initialization Methods O Hybrid Initialization Standard Initialization Compute from rotor-inlet Ŧ Reference Frame Relative to Cell Zone Absolute Initial Values Gauge Pressure (atm) 0.9000002 X Velocity (m/s) 2.518632e-05 Y Velocity (m/s) 1.39344e-06 Z Velocity (m/s) -130.9983Temperature (k) 279.4746 Initialize Reset Patch... Reset DPM Sources Reset Statistics

**Solving**  $\rightarrow$  Initialization  $\rightarrow$  Options...

- a. Select rotor-inlet from the Compute from drop-down list.
- b. Select Absolute in the Reference Frame list.
- c. Click Initialize.
- 8. Save the initial case file (axial\_comp.cas.gz).

**File**  $\rightarrow$  Write  $\rightarrow$  Case...

9. Run the calculation for one revolution of the rotor.

Solving  $\rightarrow$  Run Calculation  $\rightarrow$  Advanced...

Run Calculation			
Check Case	Preview Mesh Motion		
Time Stepping Method	Time Step Size (s)		
Fixed 🔻	6.6667e-6 P		
Settings	Number of Time Steps		
	240 🜩		
Options			
Extrapolate Variables			
🔲 Data Sampling for Tim	ne Statistics		
Sampling Interval			
1	Sampling Options		
Time Sampled	(s) 0		
Max Iterations/Time Step	Reporting Interval		
20	1		
Profile Update Interval			
1			
Data File Quantities	Acoustic Signals		
	Acoustic Sources FFT		
Calculate			
Help			

a. Enter 6.6667e-6 s for **Time Step Size**.

The time step is set such that the passing of a single rotor blade is divided into 15 time steps. There are 16 blades on the rotor. Therefore, in each time step the rotor rotates 360/16/15=1.5 degrees. With a rotational speed of 37,500 rpm (225,000 deg/sec), 1.5 degrees of rotation takes 1.5 / 2.25e5 = 6.6667e-6 sec.

b. Enter 240 for Number of Time Steps.

There are 16 blades on the rotor, and each rotor blade period corresponds to 15 time steps (see above). Therefore, a complete revolution of the rotor will take 16\*15=240 time steps.

- c. Retain the default setting of 20 for Max Iterations/Time Step.
- d. Click Calculate.

The calculation will run for approximately 4,200 iterations.

The residuals jump at the beginning of each time step and then fall at least two to three orders of magnitude. Also, the relative convergence criteria is achieved before reaching the maximum iteration limit (20) for each time step, indicating the limit does not need to be increased.

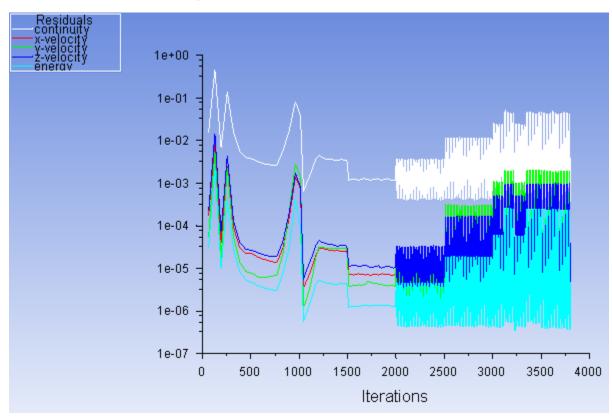


Figure 11.3: Residual History for the First Revolution of the Rotor

10. Examine the flow variable histories for the first revolution of the rotor (Figure 11.4: Mass Flow Rate at the Inlet During the First Revolution (p. 481), Figure 11.5: Mass Flow Rate at the Outlet During the First Revolution (p. 481), and Figure 11.6: Static Pressure at the Interface During the First Revolution (p. 482)).

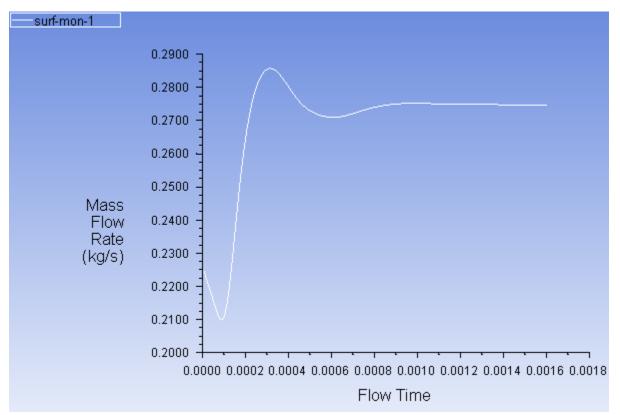
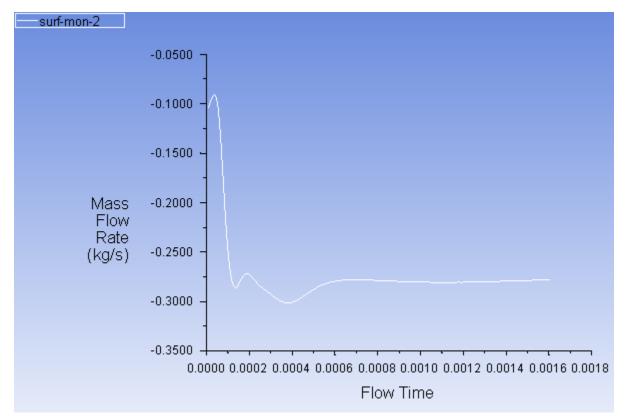
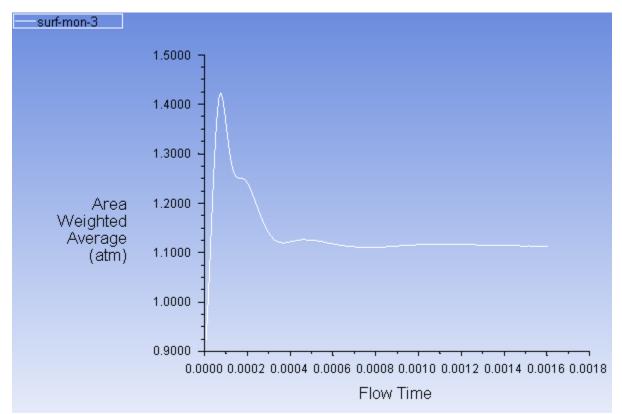


Figure 11.4: Mass Flow Rate at the Inlet During the First Revolution

Figure 11.5: Mass Flow Rate at the Outlet During the First Revolution





#### Figure 11.6: Static Pressure at the Interface During the First Revolution

The flow variable histories show that the large variations in flow rate and interface pressure that occur early in the calculation are greatly reduced as time-periodicity is approached.

11. Save the case and data files (axial\_comp-0240.cas.gz and axial\_comp-0240.dat.gz).

#### Note

It is a good practice to save the case file whenever you are saving the data file especially for sliding mesh model. This is because the case file contains the mesh information, which is changing with time.

#### Note

For transient-state calculations, you can add the character string %t to the file name so that the iteration number is automatically appended to the name (for example, by entering axial\_comp-%t for the **File Name** in the **Select File** dialog box, ANSYS Fluent will save files with the names axial\_comp-0240.cas and axial\_comp-0240.dat).

12. Rename the report output file in preparation for further iterations.

**Solution**  $\rightarrow$  Monitors  $\rightarrow$  Report Files  $\rightarrow$  surf-mon-1-rfile  $\stackrel{\frown}{\rightarrow}$  Edit...

Edit Report File	
Name	
surf-mon-1-rfile	
Available Report Definitions [0/5]	Selected Report Definitions [0/1]
delta-time flow-time iters-per-timestep surf-mon-2 surf-mon-3	Add>> < <remove< td=""></remove<>
Output File Base Name surf-mon-1b.out Browse	New   Edit
Full File Name	
Get Data Every          1       Ime-step         Ime-step       Ime-step         Ime-step       Ime-step	
NO	Cancel Help

- a. Enter surf-mon-1b.out for Output File Base Name.
- b. Click **OK** to close the **Edit Report File** dialog box.
- 13. Similarly, change the output file names for the **surf-mon-2-rfile** and **surf-mon-3-rfile** report file definitions to surf-mon-2b.out and surf-mon-3b.out, respectively.
- 14. Continue the calculation for 720 more time steps to simulate three more revolutions of the rotor.

**Solving**  $\rightarrow$  Run Calculation  $\rightarrow$  Advanced...

Run Calculation			
Check Case	Preview Mesh Motion		
Time Stepping Method	Time Step Size (s)		
Fixed 🔻	6.6667e-06		
Settings	Number of Time Steps		
	720 🚖		
Options			
Extrapolate Variables			
Data Sampling for Time	ne Statistics		
Sampling Interval			
1	Sampling Options		
Time Sampled	(s) 0		
Max Iterations/Time Step			
20 🜩	1		
Profile Update Interval			
Data File Quantities	Acoustic Signals		
	Acoustic Sources FFT		
Calculate			
Help			

#### Note

Calculating three more revolutions will require some additional CPU time. If you choose, instead of calculating the solution, you can read a data file (axial\_comp-0960.dat.gz) with the precalculated solution for this tutorial. This data file can be found in the slid-ing\_mesh directory.

The calculation will run for approximately 11,600 more iterations.

15. Examine the flow variable histories for the next three revolutions of the rotor to verify that the solution is time-periodic (Figure 11.7: Mass Flow Rate at the Inlet During the Next 3 Revolutions (p. 485) Figure 11.8: Mass Flow Rate at the Outlet During the Next 3 Revolutions (p. 486), and Figure 11.9: Static Pressure at the Interface During the Next 3 Revolutions (p. 486)).

#### Note

If you read the provided data file instead of iterating the solution for three revolutions, the flow variable histories can be displayed by using the **File XY Plot** dialog box.

**E** Results  $\rightarrow$  **\bigcirc** Plots  $\rightarrow$  **\stackrel{\frown}{\equiv}** File  $\stackrel{\frown}{\subseteq}$  Edit...

Click the **Add** button in the **File XY Plot** dialog box to select one of the flow variable histories from the **Select File** dialog box, click **OK**, and then click **Plot**. To obtain a better view of the data, you may want to manually change the ranges of the axes.

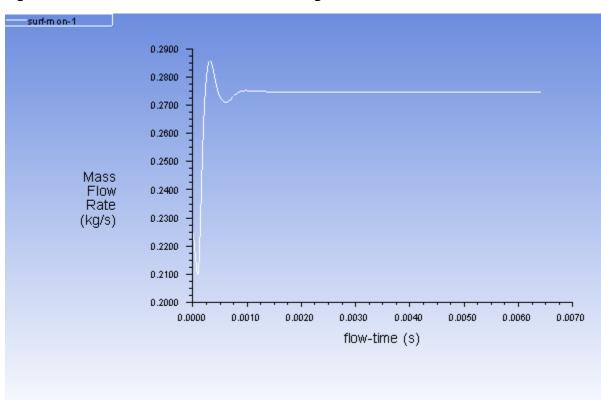


Figure 11.7: Mass Flow Rate at the Inlet During the Next 3 Revolutions

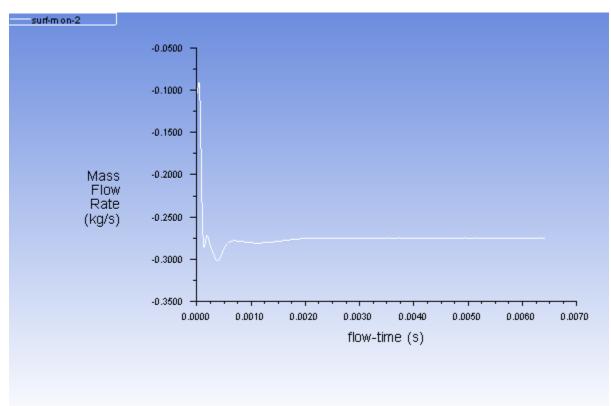
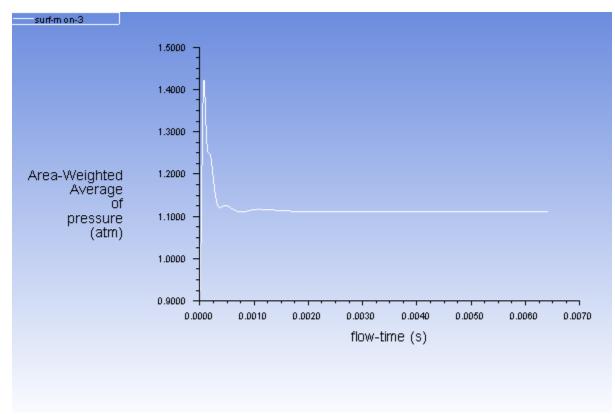


Figure 11.8: Mass Flow Rate at the Outlet During the Next 3 Revolutions





16. Save the case and data files (axial\_comp-0960.cas.gz and axial\_comp-0960.dat.gz).

**File**  $\rightarrow$  Write  $\rightarrow$  Case & Data...

- 17. Change the file names for surf-mon-1b.out, surf-mon-2b.out, and surf-mon-3b.out to surf-mon-1c.out, surf-mon-2c.out, and surf-mon-3c.out, respectively (as described in a previous step), in preparation for further iterations.
- 18. Add a point at the interface of the stator.

Postprocessing  $\rightarrow$  Surface  $\rightarrow$  Create  $\rightarrow$  Point...

Point Surface		
Options	x0 (m) -0.02	
Reset	y0 (m) -0.08	
	z0 (m) -0.036	
S	Select Point with Mouse	]
New Surface Name	e	
point-1		]
Create	anage Close Help	]

- a. Enter -0.02 for x0, -0.08 for y0, and -0.036 for z0 in the Point Surface dialog box.
- b. Retain the default, point-1 for New Surface Name.
- c. Click Create and close the Point Surface dialog box.
- 19. Enable plotting of the static pressure at a point on the stator interface (**point-1**).

**Solving**  $\rightarrow$  Reports  $\rightarrow$  Definitions  $\rightarrow$  New  $\rightarrow$  Surface Report  $\rightarrow$  Vertex Average...

Surface Report Definition	
Name	Report Type
surf-mon-4	Vertex Average 🔹
Options	Custom Vectors
	Vectors of
Per Surface	▼
Average Over	Custom Vectors
1	Field Variable
Report Files [0/3]	
surf-mon-1-rfile	Static Pressure 🔹
surf-mon-2-rfile	Surfaces point-1 🗙 📆 📆 📆
surf-mon-3-rfile	point-1
Report Plots [0/1]	]
surf-mon-3-rplot	
Create	
Report File	
Report Plot	
Frequency 1	
Print to Console	Highlight Surfaces
Create Output Parameter	New Surface 🔻
OK C	ompute Cancel Help

- a. Enter **surf-mon-4** for the **Name** of the surface report definition.
- b. In the Create group box, enable Report File, Report Plot and Print to Console.
- c. Retain the defaults of **Pressure** and **Static Pressure** for **Field Variable**.
- d. Enter point-1 in the **Surfaces** field to filter the list.
- e. Select **point-1** from the **Surfaces** selection list.
- f. Click **OK** to save the surface report definition settings and close the **Surface Report Definition** dialog box.
- 20. Continue the calculation for one final revolution of the rotor, while saving data samples for the postprocessing of the time statistics.

Solving → Run Calculation → Advanced...

Run Calculation	
Check Case	Preview Mesh Motion
Time Stepping Method	Time Step Size (s)
Fixed 🔹	6.6667e-06
Settings	Number of Time Steps
	240 🗘
Options	
Extrapolate Variables	
Data Sampling for Tir	ne Statistics
Sampling Interval	
1	Sampling Options
Time Sampled	i (s) 0
Solid Time Step	
O User Specified	
Automatic	
Max Iterations/Time Step 20	Reporting Interval
Profile Update Interval	
1	
Data File Quantities	Acoustic Signals
	Acoustic Sources FFT
Calculate	
Help	

- a. Enter 240 for Number of Time Steps.
- b. Enable Data Sampling for Time Statistics in the Options group box.

Enabling **Data Sampling for Time Statistics** causes ANSYS Fluent to calculate and store mean and root-mean-square (RMS) values of various quantities and field functions over the calculation interval.

c. Click Calculate.

The calculation will run for approximately 3,800 more iterations.

21. Save the case and data files (axial\_comp-1200.cas.gz and axial\_comp-1200.dat.gz).

**File**  $\rightarrow$  Write  $\rightarrow$  Case & Data...

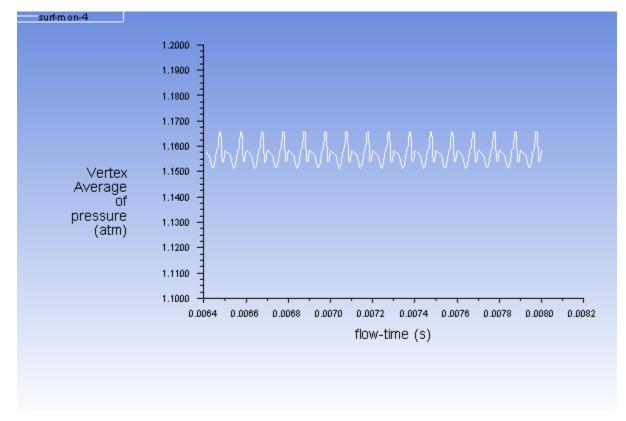


Figure 11.10: Static Pressure at a Point on The Stator Interface During the Final Revolution

### 11.4.11. Postprocessing

In the next three steps you will examine the time-averaged values for the mass flow rates at the inlet and the outlet during the final revolution of the rotor. By comparing these values, you will verify the conservation of mass on a time-averaged basis for the system over the course of one revolution.

1. Examine the time-averaged mass flow rate at the inlet during the final revolution of the rotor (as calculated from surf-mon-lc.out).



E Fourier Transform	
Options	Y Axis Function
Write FFT to File	Power Spectral Density
Acoustics Analysis	X Axis Function
	Frequency (Hz)
	Plot/Modify Input Signal
Reference Acoustic Pressure (atm)	Plot Title
1.9738e-10	Spectral Analysis of surf-mon-1-rfile
Y Axis Label	X Axis Label
Power Spectral Density	Frequency (Hz)
Y Axis Variable	X Axis Variable
surf-mon-1	Time Step 🔹
Files	Load Input File
D:/Tutorials/Sliding Mesh/surf-mon-1c.out	Free File Data
D:/Tutorials/Sliding Mesh/surf-mon-1c.out	
Plot FFT Axes	Curves Close Help

a. Click the Load Input File... button to open the Select File dialog box.

ook in: 🛛 🔒 D:\Tu	torials\Sliding Mesh		- 0 0 0	📑 🖽 🛛
Name	Size	Тŷр	Date Modified	
l solution_files		Er	25/11/21:45 PN	
획 axial_comp-0960.dat.g	<sub>JZ</sub> 1.1 MB		28/10/24:50 PN	
획 axial_comp-1200.dat.g	<sub>JZ</sub> 1.6 MB		28/10/26:38 PN	
획 axial_comp-0240.dat.o	<sub>JZ</sub> 1.1 MB		28/10/26:42 PN	
🧾 surf-mon-2b.out	11 KB		28/10/29:44 PN	
🧾 surf-mon-4-rfile.out	3 KB		28/10/25:30 PN	
🧾 surf-mon-2-rfile.out			28/10/25:04 PN	
surf-mon-1c.out	3 KB		28/10/25:30 PN	
🧃 surf-mon-3-rfile.out	3 KB		28/10/25:04 PN	
🧃 surf-mon-1b.out			28/10/29:44 PN	
surf-mon-3c.out			28/10/25:30 PN	
surf-mon-1-rfile.out			28/10/25:04 PN	
surf-mon-2c.out	3 KB		28/10/25:30 PN	
put Signal File surf-mon-	1c.out			ОК
les of type: Input Sig	nal Files (*.ard* *.A	RD*	*.xy* *.dat* *.XY* *.DAT* *.out* *hist* ) 🔻	Cancel
ter String				Filter

- i. Select **All Files** from the **files of type:** drop-down list.
- ii. Select surf-mon-1c.out from the list of files.
- iii. Click **OK** to close the **Select File** dialog box.
- b. Click the Plot/Modify Input Signal... button to open the Plot/Modify Input Signal dialog box.

Plot/Modify Input Signal		
Options Clip to Range	Signal Statistics Min	Window None
<ul> <li>Subtract Mean Value</li> <li>Subdivide into Segments</li> </ul>	0.2747385 Max	Segment Control
X Axis Range	0.274752 Mean	Samples
Min 961	0.2747454 Variance	<ul> <li>Frequency</li> <li>Samples per Segment</li> </ul>
Max 1200	2.263336e-11 Number of Samples	240 - Even (hz)
Set Defaults	240	0.0041841
	Min Frequency (hz) 0.0041841	Overlap (0 to 1) 0
Signal Plot Title		
surf-mon-1-rfile		
Y Axis Label	X Axis Label	
surf-mon-1	Time Step	
Y Axis Variable	X Axis Variab	le
surf-mon-1	▼ Time Step	▼
Apply/Plot	Axes Write (	Close Help

- i. Examine the values for Min, Max, Mean, and Variance in the Signal Statistics group box.
- ii. Close the Plot/Modify Input Signal dialog box.
- c. Select the directory path ending in **surf-mon-1c.out** from the **Files** selection list.
- d. Click the Free File Data button.
- 2. Examine the time-averaged mass flow rate at the outlet during the final revolution of the rotor (as calculated from surf-mon-2c.out), and plot the data.

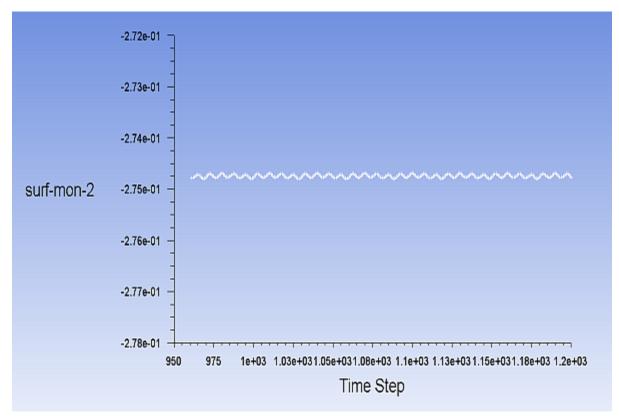
#### Postprocessing $\rightarrow$ Plots $\rightarrow$ FFT...

- a. Click the Load Input File... button to open the Select File dialog box.
  - i. Select **All Files** from the **files of type:** drop-down list.
  - ii. Select surf-mon-2c.out from the list of files.

- iii. Click **OK** to close the **Select File** dialog box.
- b. Click the **Plot/Modify Input Signal...** button to open the **Plot/Modify Input Signal** dialog box.

Plot/Modify Input Signal		<b>—</b>	
Options <ul> <li>Clip to Range</li> </ul>	Signal Statistics Min	Window None	
Subtract Mean Value	-0.2747947	Segment Control	
Subdivide into Segments	Max	Control Method	
X Axis Range	-0.274694	Samples	
Min	Mean	Frequency	
961	-0.2747451	Samples per Segment	
Max	Variance	240	
1200	9.221244e-10	Frequency Resolution (hz)	
	Number of Samples	0.0041841	
Set Defaults	240 Min Frequency (hz)	Overlap (0 to 1)	
	0.0041841		
	0.0041641	0	
Signal Plot Title			
surf-mon-2-rfile			
Y Axis Label	X Axis Label		
surf-mon-2	Time Step		
Y Axis Variable	X Axis Variab	le	
surf-mon-2	▼ Time Step	<b>~</b>	
Apply/Plot	Axes Write	Close Help	

- i. Examine the values for Min, Max, and Variance in the Signal Statistics group box.
- ii. Click Set Defaults.
- iii. Click **Apply/Plot** to display the area-weighted average of mass flow rate at the outlet (Figure 11.11: Area-Weighted Average Mass Flow Rate at the Outlet During the Final Revolution (p. 494)).



# Figure 11.11: Area-Weighted Average Mass Flow Rate at the Outlet During the Final Revolution

- iv. Close the Plot/Modify Input Signal dialog box.
- 3. Examine the vertex-averaged static pressure at the stator during the final revolution of the rotor (as calculated from surf-mon-4.out), and plot the data.

```
Postprocessing \rightarrow Plots \rightarrow FFT...
```

- a. Click the Load Input File... button to open the Select File dialog box.
  - i. Select All Files from the Files of type: drop-down list.
  - ii. Select surf-mon-4-rfile.out from the list of files.
  - iii. Click **OK** to close the **Select File** dialog box.
- b. Click the Plot/Modify Input Signal... button to open the Plot/Modify Input Signal dialog box.

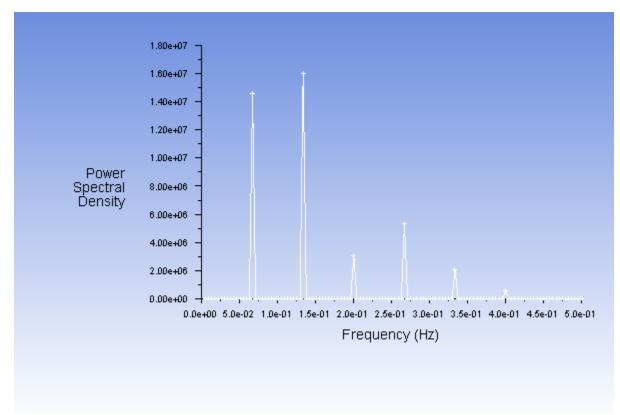
Options	Signal Statistics	Window
Clip to Range	Min	None
Subtract Mean Value	116651.9	Segment Control
Subdivide into Segments	Max	Control Method
X Axis Range	118098.8	Samples
Min	Mean	Frequency
961	117226.3	Samples per Segment
Мах	Variance	240
1200	171849.3 Number of Samples	Frequency Resolution (hz)
Set Defaults	240	0.0041841
Set Delauits	Min Frequency (hz)	Overlap (0 to 1)
	0.0041841	
	0.0041041	
Signal Plot Title		
surf-mon-4-rfile		
( Axis Label	X Axis Label	l
surf-mon-4	Time Step	
Y Axis Variable	X Axis Varial	ble
surf-mon-4	<ul> <li>Time Step</li> </ul>	

- i. Enable Subtract Mean Value in the Options group box.
- ii. Click Apply/Plot.
- iii. Close the Plot/Modify Input Signal dialog box.
- c. Click **Plot FFT** in the **Fourier Transform** dialog box.
- d. Click Axes... to open the Axes Fourier Transform dialog box.

Axes - Fourier Transform		<b>—</b> ×
Axis X Y Label	Number Format Type exponential Precision 1	Major Rules Color foreground Weight 1
Options <ul> <li>Log</li> <li>Auto Range</li> <li>Major Rules</li> <li>Minor Rules</li> </ul>	Range Minimum 0 Maximum 0 Ply Close Help	Minor Rules Color dark gray Weight 1

- e. Select **exponential** from the **Type** drop-down list, and set **Precision** to 1 in the **Number Format** group box.
- f. Click Apply and close the Axes Fourier Transform dialog box.
- g. Click Plot FFT and close the Fourier Transform dialog box.

Figure 11.12: FFT of Static Pressure at the Stator



The FFT plot clearly shows that the pressure fluctuations due to interaction at the interface are dominated by the rotor and stator blade passing frequencies (which are 10 kHz and 20 kHz, respectively) and their higher harmonics.

4. Display contours of the mean static pressure on the walls of the axial compressor.

Options Filled	Contours of Unsteady Statistics	•
Vode Values	Mean Static Pressure	-
<ul> <li>Global Range</li> <li>Auto Range</li> </ul>	Min (atm) Max (atm)	_
Clip to Range	0.2594246 1.347666	
Draw Profiles Draw Mesh	Surfaces Filter Text	X
Coloring Banded Smooth Levels Setup 20 1	<ul> <li>Wall         <ul> <li>rotor-blade-1</li> <li>rotor-blade-2</li> <li>rotor-hub</li> <li>rotor-shroud</li> <li>stator-blade-1</li> <li>stator-blade-2</li> </ul> </li> <li>New Surface          <ul> <li>Display</li> <li>Compute</li> <li>Close</li> <li>Help</li> </ul> </li> </ul>	•

#### Postprocessing $\rightarrow$ Graphics $\rightarrow$ Contours $\rightarrow$ Edit

- a. Enable **Filled** in the **Options** group box.
- b. Select Unsteady Statistics... and Mean Static Pressure from the Contours of drop-down lists.
- c. Select wall from the Surface Types selection list.

Scroll down the **Surfaces** selection list to find **wall**.

- d. Click **Display** and close the **Contours** dialog box.
- e. Rotate the view to get the display as shown in Figure 11.13: Mean Static Pressure on the Outer Shroud of the Axial Compressor (p. 498).

Shock waves are clearly visible in the flow near the outlets of the rotor and stator, as seen in the areas of rapid pressure change on the outer shroud of the axial compressor.

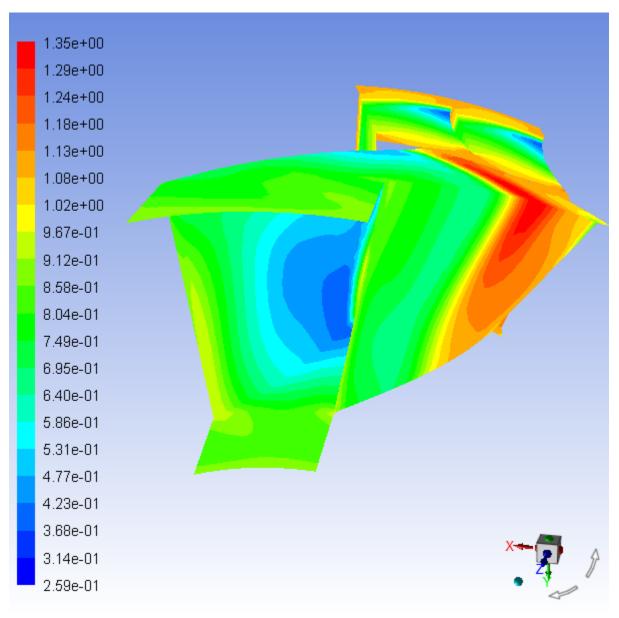


Figure 11.13: Mean Static Pressure on the Outer Shroud of the Axial Compressor

## 11.5. Summary

This tutorial has demonstrated the use of the sliding mesh model for analyzing transient rotor-stator interaction in an axial compressor stage. The model utilized the coupled pressure-based solver in conjunction with the transient algorithm to compute the inviscid flow through the compressor stage. The solution was calculated over time until the reported variables displayed time-periodicity (which required several revolutions of the rotor), after which time-averaged data was collected while running the case for the equivalent of one additional rotor revolution (240 time steps).

The Fast Fourier Transform (FFT) utility in ANSYS Fluent was employed to determine the time averages from stored flow variable report data. You also used the FFT utility to examine the frequency content of the transient report data. The observed peak corresponds to the passing frequency and the higher harmonics of the passing frequency, which occurred at approximately 10,000 Hz.

## **11.6. Further Improvements**

This tutorial guides you through the steps to reach a second-order solution. You may be able to obtain a more accurate solution by adapting the mesh. Adapting the mesh can also ensure that your solution is independent of the mesh. These steps are demonstrated in Introduction to Using ANSYS Fluent: Fluid Flow and Heat Transfer in a Mixing Elbow (p. 121).

## **Chapter 12: Using Dynamic Meshes**

This tutorial is divided into the following sections:

- 12.1. Introduction 12.2. Prerequisites 12.3. Problem Description
- 12.4. Setup and Solution
- 12.5. Summary
- 12.6. Further Improvements

## 12.1. Introduction

In ANSYS Fluent, the dynamic mesh capability is used to simulate problems with boundary motion, such as check valves and store separations. The building blocks for dynamic mesh capabilities within ANSYS Fluent are three dynamic mesh schemes, namely, smoothing, layering, and remeshing. A combination of these three schemes is used to tackle the most challenging dynamic mesh problems. However, for simple dynamic mesh problems involving linear boundary motion, the layering scheme is often sufficient. For example, flow around a check valve can be simulated using only the layering scheme, as is demonstrated in this tutorial.

Check valves are commonly used to allow unidirectional flow. For instance, they are often used to act as a pressure-relieving device by only allowing fluid to leave the domain when the pressure is higher than a certain level. In such a case, a ball is connected to a spring that acts to push the ball to the valve seat and to shut the flow. But when the pressure force on the ball is greater than the spring force, the ball will move away from the valve seat and allow fluid to leave, thus reducing the pressure upstream. Gravity could be another factor in the force balance, and can be considered in ANSYS Fluent. The deformation of the ball is typically neglected, thus allowing for a rigid body Fluid Structure Interaction (FSI) calculation, which can be handled by Fluent's six degrees of freedom (six DOF) solver.

This tutorial provides information for performing basic dynamic mesh calculations by demonstrating how to do the following:

- Use the dynamic mesh capability of ANSYS Fluent to solve a simple flow-driven rigid-body motion problem.
- Set boundary conditions for internal flow.
- Set up the rigid-body motion of the ball, by specifying a set of six DOF properties that define the translation, spring loading, and range.
- · Calculate a solution using the pressure-based solver.

## 12.2. Prerequisites

This tutorial is written with the assumption that you have completed one or more of the introductory tutorials found in this manual:

• Introduction to Using ANSYS Fluent in ANSYS Workbench: Fluid Flow and Heat Transfer in a Mixing Elbow (p. 1)

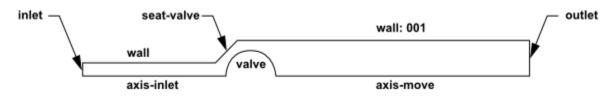
- Parametric Analysis in ANSYS Workbench Using ANSYS Fluent (p. 73)
- Introduction to Using ANSYS Fluent: Fluid Flow and Heat Transfer in a Mixing Elbow (p. 121)

and that you are familiar with the ANSYS Fluent tree and ribbon structure. Some steps in the setup and solution procedure will not be shown explicitly.

## 12.3. Problem Description

The check valve problem to be considered is shown schematically in Figure 12.1: Problem Specification (p. 502). A 2D axisymmetric valve geometry is used, consisting of a mass-flow inlet on the left, and a pressure outlet on the right, affecting the motion of a valve ball. In this case, the transient motion of the valve ball due to spring force and hydrodynamic force is studied.

#### Figure 12.1: Problem Specification



## 12.4. Setup and Solution

The following sections describe the setup and solution steps for this tutorial:

12.4.1. Preparation 12.4.2. Mesh 12.4.3. General Settings 12.4.4. Models 12.4.5. Materials 12.4.6. Boundary Conditions 12.4.7. Solution: Steady Flow 12.4.8. Time-Dependent Solution Setup 12.4.9. Mesh Motion 12.4.10. Time-Dependent Solution 12.4.11. Postprocessing

### 12.4.1. Preparation

To prepare for running this tutorial:

- 1. Set up a working folder on the computer you will be using.
- 2. Go to the ANSYS Customer Portal, https://support.ansys.com/training.

#### Note

If you do not have a login, you can request one by clicking **Customer Registration** on the log in page.

3. Enter the name of this tutorial into the search bar.

- 4. Narrow the results by using the filter on the left side of the page.
  - a. Click ANSYS Fluent under Product.
  - b. Click 18.0 under Version.
- 5. Select this tutorial from the list.
- 6. Click the **dynamic\_mesh\_R180.zip** link to download the input files.
- 7. Unzip dynamic\_mesh\_R180.zip to your working folder.

The mesh file valve.msh can be found in the dynamic\_mesh folder created after unzipping the file.

8. Use Fluent Launcher to start the **2D** version of ANSYS Fluent.

Fluent Launcher displays your **Display Options** preferences from the previous session.

- 9. Ensure that the **Display Mesh After Reading** and **Workbench Color Scheme** options are enabled.
- 10. Ensure that **Double Precision** is disabled if you want to match the tutorial setup exactly.
- 11. Select Serial under Processing Options.

If you decide to instead run this tutorial in parallel, make sure to use **Principal Axes** as the partitioning method.

For more information about the Fluent Launcher, see starting ANSYS Fluent using the Fluent Launcher in the Fluent Getting Started Guide.

### 12.4.2. Mesh

1. Read the mesh file valve.msh.

File → Read → Mesh...

### 12.4.3. General Settings

1. Check the mesh.

Setting Up Domain  $\rightarrow$  Mesh  $\rightarrow$  Check

#### Note

You should always make sure that the cell minimum volume is not negative, since ANSYS Fluent cannot begin a calculation if this is the case.

2. Change the display units for length to millimeters in the Set Units dialog box.

Setting Up Domain  $\rightarrow$  Mesh  $\rightarrow$  Units...

- a. Select length under Quantities and mm under Units.
- b. Close the **Set Units** dialog box.
- 3. Display the mesh (Figure 12.2: Initial Mesh for the Valve (p. 505)).

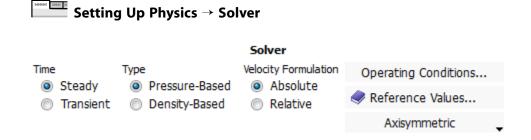
💶 Mesh Display	/		ĸ
Options Nodes	Edge Type All	Surfaces Filter Text	x
Edges	Feature	axis-inlet	1
Faces Partitions	Outline	axis-move default-interior	
Overset			Ξ
Shrink Factor	Fosturo Anglo	inlet	
	20	int-layering outlet	
Outline	Interior	seat-valve	Ŧ
Adjacency		New Surface 🔻	
Display Colors Close Help			H

- a. Deselect axis-inlet, axis-move, inlet, and outlet from the Surfaces selection list.
- b. Click **Display**.



0100(mm.)

- c. Close the Mesh Display dialog box.
- 4. Enable an axisymmetric steady-state calculation.



Select Axisymmetric from the drop-down list in the Solver group box (below Reference Values...).

### 12.4.4. Models

1. Enable the standard  $k - \varepsilon$  turbulence model.

Setting Up Physics  $\rightarrow$  Models  $\rightarrow$  Viscous...

<b>E</b> Viscous Model	<b>—</b>
Model Inviscid Laminar Spalart-Allmaras (1 eqn) k-epsilon (2 eqn) k-omega (2 eqn) Transition k-kl-omega (3 eqn) Transition SST (4 eqn) Reynolds Stress (5 eqn) Scale-Adaptive Simulation (SAS) Detached Eddy Simulation (DES)	Model Constants Cmu 0.09 C1-Epsilon 1.44 C2-Epsilon 1.92 TKE Prandtl Number 1 TDR Prandtl Number
k-epsilon Model Standard RNG Realizable Near-Wall Treatment Standard Wall Functions Scalable Wall Functions Non-Equilibrium Wall Functions Enhanced Wall Treatment Menter-Lechner User-Defined Wall Functions Enhanced Wall Treatment Options Pressure Gradient Effects	1.3 User-Defined Functions Turbulent Viscosity none Prandtl Numbers TKE Prandtl Number none TDR Prandtl Number none
Options  Production Kato-Launder  Production Limiter	Cancel Help

- a. Select **k-epsilon (2 eqn)** from the **Model** list and retain the default selection of **Standard** in the **k-epsilon Model** group box.
- b. Select Enhanced Wall Treatment for the Near-Wall Treatment.
- c. Click **OK** to close the **Viscous Model** dialog box.

## 12.4.5. Materials

1. Apply the ideal gas law for the incoming air stream.

**Setting Up Physics**  $\rightarrow$  Materials  $\rightarrow$  Create/Edit...

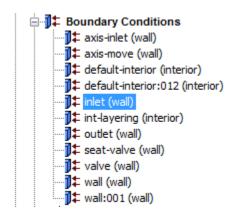
Create/Edit Materials		<b>—</b>
Name	Material Type	Order Materials by
air	fluid	
Chemical Formula	Fluent Fluid Materials	Chemical Formula
	air	Fluent Database
	Mixture	User-Defined Database
	none	User-Defined Database
Properties		
Density (kg/m3) ideal-gas	▼ Edit 🌰	
Cp (Specific Heat) (j/kg-k) constant	Edt	
	E	
1006.43		
Thermal Conductivity (w/m-k) constant	Edit	
0.0242		
Viscosity (kg/m-s) constant	<ul> <li>Edit</li> </ul>	
1.7894e-05		
<del></del>	*	
	Change/Create Delete Close Help	

- a. Ensure air is selected from the Fluent Fluid Materials drop-down list.
- b. Select ideal-gas from the Density drop-down list.
- c. Click Change/Create.
- d. Close the Create/Edit Materials dialog box.

Note that the energy equation is automatically enabled based on your change to the material density method.

### 12.4.6. Boundary Conditions

Dynamic mesh motion and all related parameters are specified using the items in the **Dynamic Mesh** task page, not through the **Boundary Conditions** task page. You will set these conditions in a later step.



1. Set the conditions for the mass-flow inlet (inlet).

Since the **inlet** boundary is assigned to a wall boundary type in the original mesh, you will need to explicitly assign the inlet boundary to a mass-flow inlet boundary type in ANSYS Fluent.

💶 Mass-Flow Ir	let	×
Zone Name		
inlet	-	
Momentum	Thermal Radiation Species DPM	Multiphase Potential UDS
	Reference Frame Absolute	▼
Mass Flo	w Specification Method Mass Flow Rate	▼
	Mass Flow Rate (kg/s) 0.0116	constant 🔹
Supersonic/Init	ial Gauge Pressure (pascal) 0	constant 🔹
Directio	n Specification Method Normal to Boundary	•
	Turbulence	
	Specification Method Intensity and Hydraulic	: Diameter 🔹 🔻
	Turbulent Intensity	/ (%) 5 P
	Hydraulic Diameter	(mm) 20
	OK Cancel Help	]

**T** Setup  $\rightarrow$  Boundary Conditions  $\rightarrow$  inlet  $\stackrel{\bigcirc}{\rightarrow}$  Type  $\rightarrow$  mass-flow-inlet

- a. Enter 0.0116 kg/s for Mass Flow Rate.
- b. Select Normal to Boundary from the Direction Specification Method drop-down list.
- c. Select **Intensity and Hydraulic Diameter** from the **Specification Method** drop-down list in the **Tur-bulence** group box.
- d. Retain 5% for Turbulent Intensity.
- e. Enter 20 mm for the Hydraulic Diameter.
- f. Click OK to close the Mass-Flow Inlet dialog box.
- 2. Set the conditions for the exit boundary (outlet).

Since the **outlet** boundary is assigned to a wall boundary type in the original mesh, you will need to explicitly assign the outlet boundary to a pressure outlet boundary type in ANSYS Fluent.

**E** Setup  $\rightarrow$  Boundary Conditions  $\rightarrow$  outlet  $\stackrel{\textcircled{}}{\rightarrow}$  Type  $\rightarrow$  pressure-outlet

Pressure Outlet					×
Zone Name					
outlet					
Momentum	Thermal Radiation Species	DPM Multip	hase Po	otential U	DS
Back	flow Reference Frame Absolute				•
	Gauge Pressure (pascal) 0		consta	nt	-
Backflow Direction	n Specification Method From Neigh	boring Cell			-
Backflow	Pressure Specification Total Press	ure			-
Average Press	ure Specification				
🔲 Target Mass Flo	ow Rate				
	Turbulence				
	Specification Method Intensity an	d Hydraulic Diame	eter		•
	Backflow Turbuler	nt Intensity (%)	5		P
	Backflow Hydraulic	Diameter (mm)	50		P
	OK	el Help			

- a. Select From Neighboring Cell from the Backflow Direction Specification Method drop-down list.
- b. Select **Intensity and Hydraulic Diameter** from the **Specification Method** drop-down list in the **Turbulence** group box.
- c. Retain 5% for **Backflow Turbulent Intensity**.
- d. Enter 50 mm for Backflow Hydraulic Diameter.
- e. Click OK to close the Pressure Outlet dialog box.
- 3. Set the boundary type to **axis** for the **axis-inlet** boundary.

Since the **axis-inlet** boundary is assigned to a wall boundary type in the original mesh, you will need to explicitly assign this boundary to an axis boundary type in ANSYS Fluent.

**Example :** Setup  $\rightarrow$  Boundary Conditions  $\rightarrow$  axis-inlet  $\stackrel{\frown}{\rightarrow}$  Type  $\rightarrow$  axis

4. Set the boundary type to **axis** for the **axis-move** boundary.



### 12.4.7. Solution: Steady Flow

In this step, you will generate a steady-state flow solution that will be used as an initial condition for the time-dependent solution.

1. Set the solution parameters.

Solution Methods	
Pressure-Velocity Coupling	
Scheme	
Coupled	•
Spatial Discretization	_
Gradient	1
Least Squares Cell Based 🔹	
Pressure	
PRESTO!	Ξ
Density	
Second Order Upwind 👻	
Momentum	
Second Order Upwind 👻	
Turbulent Kinetic Energy	
First Order Upwind 👻	_
Transient Formulation	
Non-Iterative Time Advancement	
Frozen Flux Formulation	
Pseudo Transient	
Warped-Face Gradient Correction	
High Order Term Relaxation Options	
Default	

**Solving**  $\rightarrow$  Solution  $\rightarrow$  Methods...

- a. Select **Coupled** from the **Scheme** drop-down list.
- b. Select **PRESTO!** from the **Pressure** drop-down list.
- c. Retain the default of Second Order Upwind in the Density drop-down list.
- d. Retain the default of Second Order Upwind in the Momentum drop-down list.
- e. Retain the defaults of First Order Upwind in the Turbulent Kinetic Energy and Turbulent Dissipation Rate drop-down lists.
- f. Retain the default of Second Order Upwind in the Energy drop-down list.
- 2. Set the relaxation factors.



Solutio	on Controls		
Flow Co	ourant Numbe	er	
200			
	– Explicit Rela	axation Factors	
	Momentum	0.5	
	Pressure	0.5	
Under-F	Relaxation Fac	tors	
Dens	sity		
1			
Body	/ Forces		
1			
Turb	ulent Kinetic	Energy	Ξ
0.8			
Turb	ulent Dissipat	tion Rate	
0.8			
Turb	ulent Viscosit	ty .	
1			
			- <b>-</b>
Defau	t		
Equati	ons	Advanced	

Retain the default values for **Under-Relaxation Factors** in the **Solution Controls** task page.

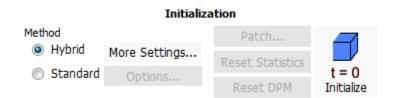
3. Enable the plotting of residuals during the calculation.

Solving $\rightarrow$ Report	rts → Residuals
------------------------------	-----------------

Residual Monitors					×
Options	Equations				
Print to Console	Residual	Monito	r Check Converge	ence Absolute Criteria	<u> </u>
V Plot	continuity	V	<b>V</b>	0.001	E
Window	x-velocity		1	0.001	
1 Curves Axes	y-velocity	V	<b>V</b>	0.001	
Iterations to Plot	energy		V	1e-06	
1000 ≑		-	[mail		
1000	Residual Values			Convergence Criterio	n
	Normalize		Iterations	absolute	
Iterations to Store			5 🗼		
1000 ≑	Scale			Convergence Condi	tions
	Compute Loca	I Scale			
OK Plot Renormalize Cancel Help					

- a. Ensure that **Plot** is enabled in the **Options** group box.
- b. Click OK to close the Residual Monitors dialog box.
- 4. Initialize the solution.

Solving → Initialization



- a. Retain the default selection of Hybrid from the Method list.
- b. Click Initialize.

#### Note

A warning is displayed in the console stating that the convergence tolerance of 1.000000e-06 was not reached during hybrid initialization. This means that the default number of iterations is not enough. You will increase the number of iterations and re-initialize the flow. For more information refer to hybrid initialization in the Fluent User's Guide.

#### c. Click More Settings....

Hybrid Initialization	n	×
General Settings	Turbulence Settings	Species Settings
	axation Factor	
Maintain Const	ant Velocity Magnitude	
	OK Cancel Help	

- i. Increase the Number of Iterations to 20.
- ii. Click **OK** to close the **Hybrid Initialization** dialog box.
- d. Click Initialize again.

#### Note

Click **OK** in the **Question** dialog box that asks if it is OK to discard the current data. The console will then display that hybrid initialization is done.

#### Note

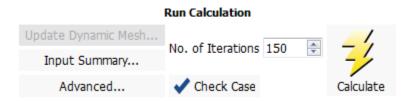
For flows in complex topologies, hybrid initialization will provide better initial velocity and pressure fields than standard initialization. This will help to improve the convergence behavior of the solver.

5. Save the case file (valve\_init.cas.gz).



6. Start the calculation by requesting 150 iterations.





- a. Enter 150 for **No. of Iterations**.
- b. Click **Calculate** to start the steady flow simulation.

The solution converges in approximately 132 iterations.

7. Save the case and data files (valve\_init.cas.gz and valve\_init.dat.gz).

File → Write → Case & Data...

### 12.4.8. Time-Dependent Solution Setup

1. Enable a time-dependent calculation.



		Solver	
Time	Type	Velocity Formulation	Operating Conditions
<ul> <li>Steady</li> <li>Transient</li> </ul>	<ul> <li>Pressure-Based</li> <li>Density-Based</li> </ul>	<ul> <li>Absolute</li> <li>Relative</li> </ul>	🥏 Reference Values
			Axisymmetric

Select Transient from the Time list.

### 12.4.9. Mesh Motion

1. Enable dynamic mesh motion and define the layering settings.

HODAL HINK IN	Setting	Up Domain	→ Mesh	Models $\rightarrow$	• Dynamic	Mesh
---------------	---------	-----------	--------	----------------------	-----------	------

Task Page		×
Dynamic Mesh Dynamic Mesh Mesh Methods Smoothing Layering Remeshing Settings	Options In-Cylinder Six DOF Implicit Update Contact Detection	
Events Dynamic Mesh Zon	es	
Create/Edit Display Zone Moti Preview Mesh Mot		
Help		

a. Enable **Dynamic Mesh** in the **Dynamic Mesh** task page.

For more information on the available models for moving and deforming zones, see the Fluent User's Guide.

b. Disable **Smoothing** and enable **Layering** in the **Mesh Methods** group box.

#### Note

ANSYS Fluent will automatically flag the existing mesh zones for use of the different dynamic mesh methods where applicable.

c. Click the **Settings...** button in the **Mesh Methods** group box to open the **Mesh Method Settings** dialog box.

🖸 Mesh Metho	od Settings	<b>X</b>
Smoothing	Layering	Remeshing
Options Options Height Ba Ratio Bas		
Split Facto Collapse Facto		
ОК	Cancel	Help

- i. Select Ratio Based in the Options group box.
- ii. Retain the default settings of 0.4 and 0.2 for **Split Factor** and **Collapse Factor**, respectively.
- iii. Click **OK** to close the **Mesh Method Settings** dialog box.
- 2. Enable the six degrees of freedom (six DOF) solver and define a set of motion properties.

The six DOF solver computes external forces and moments (such as hydrodynamic forces) on objects that undergo rigid body motion. These forces are computed by numerical integration of pressure and shear stress over the object's surfaces. Additional load forces can be added, such as those produced by a spring.

- a. Enable Six DOF in the Options group box of the Dynamic Mesh task page.
- b. Click the **Settings...** button in the **Options** group box to open the **Options** dialog box.

Options		<b>—</b>		
In-Cylinder Six DOF	Implicit Update	Contact Detection		
Six DOF Properties				
1dof				
Create/Edit Delete Delete All				
Gravitational Acceleration	n/s2) 0			
Write Motion History				
File Name				
motion				
ОК	Cancel Help			

i. Click the **Create/Edit...** button in the **Six DOF** tab of the **Options** dialog box, in order to open the **Six DOF Properties** group box.

Here you will define a set of properties for the motion of the valve ball using the ANSYS Fluent graphical user interface. Note that you could instead define it through a user-defined function; this is not as easy to do, but may be necessary for more complex motion.

Six DOF Pro	operties	×		
Name				
1dof				
Mass (kg)	V One D	OF Translation		
0.2	📃 One D	OF Rotation		
One DOF Direction				
х	Y Z			
1	0 0			
Center of R	lotation			
X (mm)	Y (mm) Z (	mm)		
0	0 0			
Spring Preload (n) -21	Constant (n/ 1000	m)		
Constraine				
-Reference	Point m) Minimum (mm)	Maximum (mm)		
10	1	185		
Inertia Tensor				
	Iyy (kg-m2) Izz (	kg-m2)		
0	0			
	Ixz (kg-m2) Iyz (	(kg-m2)		
0	0 0			
G	Create Close	Help		

- ii. Enter 1dof for Name.
- iii. Enter 0.2 for Mass.

This represents the total mass of the valve ball.

- iv. Enable the **One DOF Translation** option to ensure the valve ball only translates in a single direction without rotation.
- v. Enter 1 for **X** and 0 for **Y** in the **Direction** group box.
- vi. Enter -21 for **Preload** and 1000 for **Constant** in the **Spring** group box, in order to define a Hooke's law spring that exerts loads on the valve ball.

The **Preload** field allows you to specify that the spring is deformed (that is, extended or compressed) with the current position of the valve ball. A negative value is used in this case in accordance with the defined **Direction**, that is, to specify that the initial force of the spring acts to move the valve ball toward the valve seat. vii. Enable the **Constrained** option, so that you can prevent the valve ball from contacting the valve seat.

The valve cannot be allowed to completely close: because dynamic mesh problems require that at least one layer of cells remains in order to maintain the topology, a small gap must be maintained between the valve ball and the valve seat.

viii.Enter the following values in the Reference Point group box.

Property	Value
Location	10 mm
Minimum	1 mm
Maximum	185 mm

The **Location** assigns a coordinate value to a point located on the dynamic object (the valve ball) in its initial position. The value of the **Location** is arbitrary, and allows you to define the range using values that are most convenient for your problem. In this case, the surface of the valve ball is initially 10 mm away from the valve seat; the **Minimum** is defined relative to the **Location** and the values you entered here allow the valve ball to move only 9 mm in the direction opposite to the **Direction**.

- ix. Click Create to save your settings.
- x. Close the **Six DOF Properties** dialog box.
- xi. Enable the Write Motion History option in the Options dialog box, and enter motion for the File Name.

This ensures that a file is created during the simulation that lists the location and orientation of the dynamic object at every time step, which could be used for postprocessing.

- xii. Click **OK** to close the **Options** dialog box.
- 3. Specify the dynamic mesh zone motion for the fluid cell zone in which the valve ball moves (fluid-move).
  - a. Click Create/Edit... in the Dynamic Mesh task page, to open the Dynamic Mesh Zones dialog box.

🖸 Dynamic Mesh Zone	s					×
Dynamic Mesh Zone: Zone Names fluid-move Type Stationary Stationary Rigid Body Deforming User-Defined System Coupling	s •	Dynamic Mesh Zone	5			×
Motion Attributes Six DOF UDF/Propertie 1dof	Geometry Definition s	Meshing Option Relative Motion On Relative Zone	s Solver O		Six DOF On Passive	
Center of X (mm) 6 Y (mm) 0			Rigid Body C Theta (deg) Axis_Z	0		
	ravity Velocity		Rigid Body Angul Dmega_Z (rad/s)	ar Velocity		
Orientation Calculator.	Create Draw	Delete All Dele	ete Close H	elp		

- b. Select **fluid-move** from the **Zone Names** drop-down list.
- c. Retain the default selection of **Rigid Body** from the **Type** list.
- d. Ensure that the **On** option is enabled in the **Six DOF** group box of the **Motion Attributes** tab.
- e. Enable the **Passive** option in the **Six DOF** group box.

The only reason you are having this cell zone move is to preserve the mesh that surrounds the moving valve ball. Enabling the **Passive** option ensures that the forces on this zone do not affect the motion of the valve ball.

- f. Ensure that **1dof** is selected from the **Six DOF UDF/Properties** drop-down list, to apply the motion properties you defined previously.
- g. Enter 61 mm for **X** and 0 mm for **Y** in the **Center of Gravity Location**.

These coordinates correspond to the center of the valve ball in the initial position. These fields will be updated by ANSYS Fluent during the simulation, allowing you to track the location of this zone from within the **Dynamic Mesh Zones** dialog box.

- h. Retain the default settings of 0 for the fields in the **Center of Gravity Velocity**, **Rigid Body Orientation**, and **Rigid Body Angular Velocity** group boxes.
- i. Click **Create**.
- 4. Specify the meshing options for the boundary zone named **int-layering** in the **Dynamic Mesh Zones** dialog box. This boundary zone acts as a layering interface between the stationary inlet cell zone and the cell zone in which the valve ball moves.

1	Dynamic Mesh Zones		<b>—</b>
	Zone Names	Dynamic Mesh Zones	
	int-layering	fluid-move	
	Type Stationary		
	<ul> <li>Rigid Body</li> </ul>		
	Deforming		
	O User-Defined		
	System Coupling		
	Motion Attributes Geometry Defin	nition Meshing Options Se	olver Options
	Adjacent Zone fluid-move	Cell Height (mm) 0.5	constant 🔹
	Adjacent Zone fluid-inlet	Cell Height (mm) 0	constant 🔹
	Create	Draw Delete All Delete Clos	se Help

- a. Select int-layering from the Zone Names drop-down list.
- b. Select **Stationary** from the **Type** list.
- c. Enter 0.5 mm for the Cell Height of the fluid-move Adjacent Zone.

- d. Retain the default value of 0 mm for the **Cell Height** of the fluid-inlet **Adjacent Zone**.
- e. Click **Create**.
- 5. Specify the meshing options for the boundary zone named **outlet** in the **Dynamic Mesh Zones** dialog box. This boundary zone acts as a stationary outlet for the cell zone in which the valve ball moves.
  - a. Select outlet from the Zone Names drop-down list.
  - b. Retain the previous selection of **Stationary** from the **Type** list.
  - c. Enter 1.9 mm for the Cell Height of the fluid-move Adjacent Zone.
  - d. Click Create.
- 6. Specify the meshing options for the boundary zone named **seat-valve** in the **Dynamic Mesh Zones** dialog box. This boundary zone acts as the stationary valve seat.
  - a. Select **seat-valve** from the **Zone Names** drop-down list.
  - b. Retain the previous selection of **Stationary** from the **Type** list.
  - c. Enter 0.5 mm for the Cell Height of the fluid-move Adjacent Zone.
  - d. Click Create.
- 7. Specify the motion of the boundary zone named **valve** in the **Dynamic Mesh Zones** dialog box. This boundary zone surrounds the valve ball that moves relative to the valve seat.
  - a. Select valve from the Zone Names drop-down list.
  - b. Select **Rigid Body** from the **Type** list.
  - c. Click the **Motion Attributes** tab.
    - i. Enable the **On** option in the **Six DOF** group box.
    - ii. Ensure that the **Passive** option is disabled in the **Six DOF** group box.

This ensures that the forces on this zone affect the motion of the valve ball.

- iii. Select **1dof** from the **Six DOF UDF/Properties** drop-down list to apply the motion properties you defined previously.
- iv. Enter 61 mm for **X** and 0 mm for **Y** in the **Center of Gravity Location**.

Again, these coordinates correspond to the center of the valve ball in the initial position.

- v. Retain the default settings of 0 for the fields in the **Center of Gravity Velocity**, **Rigid Body Orientation**, and **Rigid Body Angular Velocity** group boxes.
- d. Click the **Meshing Options** tab and enter 0 mm for the **Cell Height** of the fluid-move **Adjacent Zone**.
- e. Click Create and close the Dynamic Mesh Zones dialog box.

In many moving / deforming mesh (MDM) problems, you may want to preview the mesh motion before proceeding. In this problem, the mesh motion is affected by the pressure exerted by the fluid on the valve ball and acting against the inertia of the valve ball. Hence, for this problem, mesh motion in the absence of a flow field solution is meaningless, and you will not use this feature here.

### 12.4.10. Time-Dependent Solution

1. Set the solution parameters.



Task Page ×		
Solution Methods		
Pressure-Velocity Coupling		
Scheme		
PISO		
Skewness Correction		
0		
Neighbor Correction		
1		
Skewness-Neighbor Coupling		
Spatial Discretization		
Gradient		
Least Squares Cell Based		
Pressure		
PRESTO!		
Density		
Second Order Upwind		
Momentum		
Second Order Upwind		
Turbulent Kinetic Energy		
First Order Upwind		
Transient Formulation		
First Order Implicit		
Non-Iterative Time Advancement		
Frozen Flux Formulation		
Warped-Face Gradient Correction		
High Order Term Relaxation Options		
Default		
Help		

a. Select PISO from the Scheme drop-down list in Pressure-Velocity Coupling group box.

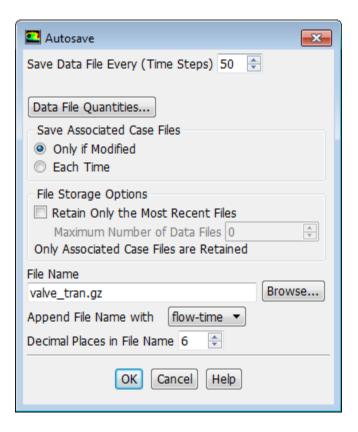
- b. Enter 0 for Skewness Correction.
- c. Retain all of the other previously set schemes and defaults.
- 2. Set the relaxation factors.

sk P	)age	
Solu	ition Controls	
ļ	Under-Relaxation Factors	
	Pressure	-
	0.6	
	Density	
	1	Ξ
	Body Forces	
	1	
	Momentum	
	0.7	
	Turbulent Kinetic Energy	
	0.4	
Def	Fault	•
_	Jations) Limits Advanced	
He	elp	

**Solving**  $\rightarrow$  Controls  $\rightarrow$  Controls...

- a. Enter 0.6 for Pressure in the Under-Relaxation Factors group box.
- b. Enter 0.4 for Turbulent Kinetic Energy.
- c. Scroll down and enter 0.4 for Turbulent Dissipation Rate.
- 3. Request that case and data files are automatically saved every 50 time steps.

**Solving**  $\rightarrow$  Activities  $\rightarrow$  Autosave...



- a. Enter 50 for Save Data File Every (Time Steps).
- b. Enter valve\_tran.gz in the File Name field.
- c. Select flow-time from the Append File Name with drop-down list.

When ANSYS Fluent saves a file, it will append the flow time value to the file name prefix (valve\_tran). The gzipped standard extensions (.cas.gz and .dat.gz) will also be appended.

- d. Click OK to close the Autosave dialog box.
- 4. Create an animation sequence for the static pressure contour plots for the valve.

Use the solution animation feature to save pressure contour plots every five time steps. After the calculation is complete, you will use the solution animation playback feature to view the animated plots over time.

Solving  $\rightarrow$  Activities  $\rightarrow$  Create  $\rightarrow$  Solution Animations...

Animation Definition				
Name: pressure				
Record after every 5 🚔 Time Step 🔻				
Storage Type HSF File				
Storage Directory als/dynamic_mesh				
Window Id 2				
Animation Object				
residuals				
contour-pressure				
New Object 🔹 Edit Object				
Save Close Help				

- a. Enter pressure for Name in the Animation Definition dialog box.
- b. Set **Record after every** to 5 and select **Time Step** from the neighboring drop-down list.

The default value of 1 instructs ANSYS Fluent to update the animation sequence at every time step. For this case, this would generate a large number of files.

c. Retain the default value of 2 for **Window Id**.

This is where the animation frames will be displayed during the calculation.

d. Click the **New Object** drop-down list and select **Contours...** from the menu that opens.

Contours				
Contour Name				
contour-pressure				
Options	Contours of			
Filled	Pressure			
Node Values	Static Pressure 🔹			
<ul> <li>Global Range</li> <li>Auto Range</li> </ul>	Min (pascal) Max (pascal)			
Clip to Range	-376.1515 671.4955			
Draw Profiles	Surfaces Filter Text			
Coloring Banded Smooth Colormap Options	outlet seat-valve valve wall wall:001 zone-surface-12			
Save/Display Compute Close Help				

- i. Enter contour-pressure for **Contour Name** in the **Contours** dialog box.
- ii. Enable Filled in the Options group box.
- iii. Retain the default selection of **Pressure...** and **Static Pressure** from the **Contours of** drop-down lists.
- iv. Click Save/Display (Figure 12.3: Contours of Static Pressure at t=0 s (p. 527)).
- v. Close the **Contours** dialog box.

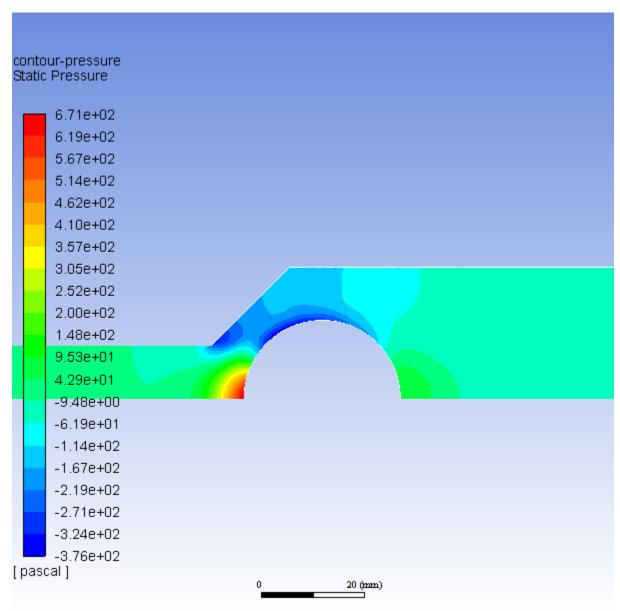


Figure 12.3: Contours of Static Pressure at t=0 s

- e. Select contour-pressure from the Animation Object selection list.
- f. Click **Save** and close the **Animation Definition** dialog box.
- 5. Create an animation sequence for the velocity vectors plots for the valve.

**Solving**  $\rightarrow$  Activities  $\rightarrow$  Create  $\rightarrow$  Solution Animations...

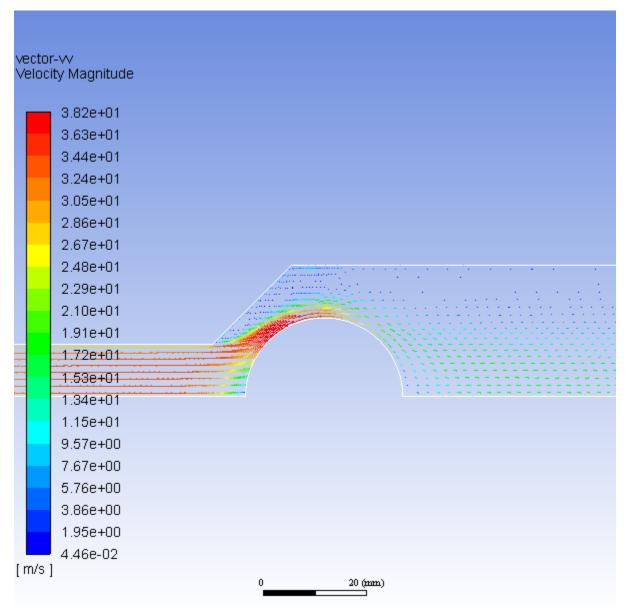
Animation Definition					
Name: vv					
Record after every 5 🚔 Time Step 🔻					
Storage Type HSF File					
Storage Directory als/dynamic_mesh					
Window Id 3					
Animation Object					
residuals					
contour-pressure vector-vv					
New Object 🔻 Edit Object					
Save Close Help					

- a. Enter vv for Name.
- b. Set **Record after every** to 5 and select **Time Step** from the neighboring drop-down list.
- c. Retain the default value of 3 for **Window Id**.
- d. Click the **New Object** drop-down list and select **Vectors...** from the menu that opens.

Vectors				
Vector Name				
vector-vv				
Options	Vectors of			
Global Range	Velocity			
Auto Range	Color by			
Clip to Range	Velocity 🔻			
Auto Scale Draw Mesh	Velocity Magnitude 🗸			
	Min Max			
Style	0 0			
arrow				
Scale Skip	Surfaces Filter Text			
1 0 🚔	axis-inlet			
Vector Options	axis-move			
Custom Vectors	derault-interior			
	default-interior:012 inlet			
Colormap Options	int-layering +			
New Surface 🔻				
Save/Display Compute Close Help				

- i. Enter vector-vv for Name.
- ii. Retain all the other default settings.
- iii. Click **Save/Display** (Figure 12.4: Vectors of Velocity at t=0 s (p. 530)).
- iv. Close the **Vectors** dialog box.





- e. Select **vector-vv** from the **Animation Object** selection list.
- f. Click Save and close the Animation Definition dialog box.

Note that the new animation definitions appear under the **Solution/Calculation Activities/Solution Animations** tree branch. To edit an animation definition, right-click it and select **Edit...** from the menu that opens.

6. Set the time step parameters for the calculation.



Task Page		
Run Calculation		
Check Case	Preview Mesh Motion	
Time Stepping Method	Time Step Size (s)	
Fixed 🔻	0.0001 P	
Settings	Number of Time Steps	
	150 🗘	
Options		
Extrapolate Variables		
Data Sampling for Tir	me Statistics	
Sampling Interval		
1	Sampling Options	
Time Sampled	d (s) 0	
Solid Time Step		
O User Specified		
Automatic		
Max Iterations/Time Step	o Reporting Interval	
20 🜩	1	
Profile Update Interval		
1		
Data File Quantities	Acoustic Signals	
Calculate		
Help		

- a. Enter 0.0001 s for **Time Step Size**.
- b. Enter 150 for Number of Time Steps.
- c. Retain 20 for Max Iterations/Time Step.

In the accurate solution of a real-life time-dependent CFD problem, it is important to make sure that the solution converges at every time step to within the desired accuracy. Here, the first few time steps will only come to a reasonably converged solution.

×

7. Save the initial case and data files for this transient problem (valve\_tran-0.000000.cas.gz and valve\_tran-0.000000.dat.gz).

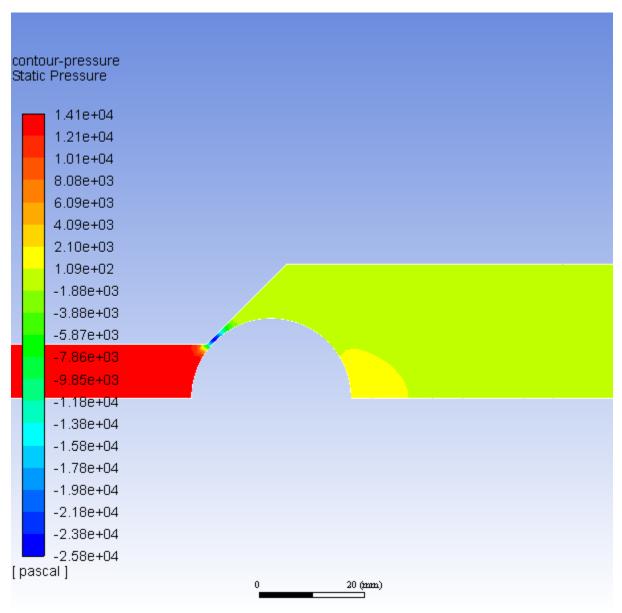
**File**  $\rightarrow$  Write  $\rightarrow$  Case & Data...

8. Calculate a solution by clicking Calculate in the Run Calculation task page.

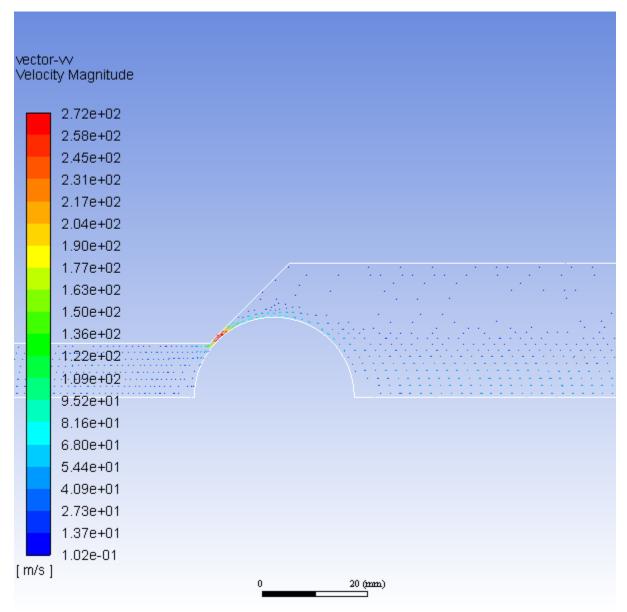
# 12.4.11. Postprocessing

- 1. Inspect the solution at the final time step.
  - a. Inspect the contours of static pressure in the valve, by clicking the **Contours of Static Pressure** tab in the graphics window (Figure 12.5: Contours of Static Pressure After 150 Time Steps (p. 532)).

Figure 12.5: Contours of Static Pressure After 150 Time Steps



b. Inspect the velocity vectors near the point where the valve ball meets the valve seat, by clicking the **Velocity Vectors Colored By Velocity Magnitude** tab in the graphics window (Figure 12.6: Velocity Vectors After 150 Time Steps (p. 533)).



#### Figure 12.6: Velocity Vectors After 150 Time Steps

2. Play the animation of the pressure contours.

Postprocessing  $\rightarrow$  Animation  $\rightarrow$  Solution Playback...

*				
Playback				
Playback	Animation Sequences			
Playback Mode Play Once 🔹	pressure			
Start Frame Increment End Frame	w			
1 🗘 1 🗘 30 🜩				
$\checkmark$				
Frame				
▲ ►				
SlowFast	Delete Delete All			
Write/Record Format Animation Frames	Picture Options			
Write Read Close Help				

a. Select **pressure** from the **Animation Sequences** list in the **Playback** dialog box.

If the Animation Sequences list is empty, click Read... to select the pressure.cxa sequence file from your working folder.

The playback control buttons will become active.

- b. Set the slider bar above Replay Speed about halfway between Slow and Fast.
- c. Retain the default settings in the rest of the dialog box and click the *button*.

You may have to change the window to see the animation. Click the second tab in the graphics window, which corresponds to the **Window Id** that you set for pressure in the **Animation Definition** dialog box.

- 3. Play the animation of the velocity vectors.
  - a. Select vv from the Animation Sequences list in the Playback dialog box.

If the Animation Sequences list does not contain vv, click Read... to select the vv. cxa sequence file from your working folder.

b. Retain the settings in the rest of the dialog box and click the 🗾 button.

You may have to change the window to see the animation. Click the third tab in the graphics window, which corresponds to the **Window Id** that you set for vv in the **Animation Definition** dialog box.

For additional information on animating the solution, see Modeling Transient Compressible Flow (p. 267) and see the Fluent User's Guide.

c. Close the **Playback** dialog box.

- 4. You can also inspect the solution at different intermediate time steps.
  - a. Read the corresponding case and data files (for example, valve\_tran-1-0.010000.cas.gz and valve\_tran-1-0.010000.dat.gz).



b. Display the desired contours and vectors.

## 12.5. Summary

In this tutorial, a check valve is used to demonstrate the dynamic layering capability within ANSYS Fluent, using one of the three dynamic mesh schemes available. You were also shown how to perform a one degree of freedom (one DOF) rigid body FSI by means of the six DOF solver.

## **12.6. Further Improvements**

This tutorial guides you through the steps to generate an initial first-order solution. You may be able to increase the accuracy of the solution further by using an appropriate higher-order discretization scheme. For a more accurate solution, you can increase the number of layers across the valve seat area. This can be achieved either by using a finer mesh at the valve seat area and/or using a non-constant layer height instead of a constant layer height, as demonstrated in this tutorial.

# **Chapter 13: Modeling Species Transport and Gaseous Combustion**

This tutorial is divided into the following sections:

13.1. Introduction
13.2. Prerequisites
13.3. Problem Description
13.4. Background
13.5. Setup and Solution
13.6. Summary
13.7. Further Improvements

# 13.1. Introduction

This tutorial examines the mixing of chemical species and the combustion of a gaseous fuel.

A cylindrical combustor burning methane ( $CH_4$ ) in air is studied using the eddy-dissipation model in ANSYS Fluent.

This tutorial demonstrates how to do the following:

- Enable physical models, select material properties, and define boundary conditions for a turbulent flow with chemical species mixing and reaction.
- Initiate and solve the combustion simulation using the pressure-based solver.
- Examine the reacting flow results using graphics.
- Predict thermal and prompt NOx production.
- Use custom field functions to compute NO parts per million.

## 13.2. Prerequisites

This tutorial is written with the assumption that you have completed one or more of the introductory tutorials found in this manual:

- Introduction to Using ANSYS Fluent in ANSYS Workbench: Fluid Flow and Heat Transfer in a Mixing Elbow (p. 1)
- Parametric Analysis in ANSYS Workbench Using ANSYS Fluent (p. 73)
- Introduction to Using ANSYS Fluent: Fluid Flow and Heat Transfer in a Mixing Elbow (p. 121)

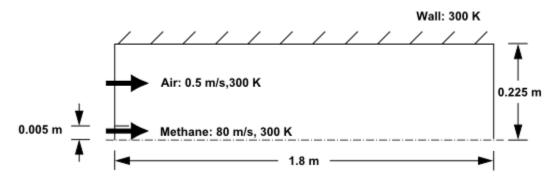
and that you are familiar with the ANSYS Fluent tree and ribbon structure. Some steps in the setup and solution procedure will not be shown explicitly.

To learn more about chemical reaction modeling, see the Fluent User's Guide and the Fluent Theory Guide. Otherwise, no previous experience with chemical reaction or combustion modeling is assumed.

# **13.3. Problem Description**

The cylindrical combustor considered in this tutorial is shown in Figure 13.1: Combustion of Methane Gas in a Turbulent Diffusion Flame Furnace (p. 538). The flame considered is a turbulent diffusion flame. A small nozzle in the center of the combustor introduces methane at 80 m/s. Ambient air enters the combustor coaxially at 0.5 m/s. The overall equivalence ratio is approximately 0.76 (approximately 28 % excess air). The high-speed methane jet initially expands with little interference from the outer wall, and entrains and mixes with the low-speed air. The Reynolds number based on the methane jet diameter is approximately  $5.7 \times 10^3$ .





# 13.4. Background

In this tutorial, you will use the generalized eddy-dissipation model to analyze the methane-air combustion system. The combustion will be modeled using a global one-step reaction mechanism, assuming complete conversion of the fuel to  $CO_2$  and  $H_2O$ . The reaction equation is  $CH_4+2O_2 \rightarrow CO_2+2H_2O$  (13.1)

This reaction will be defined in terms of stoichiometric coefficients, formation enthalpies, and parameters that control the reaction rate. The reaction rate will be determined assuming that turbulent mixing is the rate-limiting process, with the turbulence-chemistry interaction modeled using the eddy-dissipation model.

## 13.5. Setup and Solution

The following sections describe the setup and solution steps for this tutorial:

13.5.1. Preparation
13.5.2. Mesh
13.5.3. General Settings
13.5.4. Models
13.5.5. Materials
13.5.6. Boundary Conditions
13.5.7. Initial Reaction Solution
13.5.8. Postprocessing
13.5.9. NOx Prediction

## 13.5.1. Preparation

To prepare for running this tutorial:

- 1. Set up a working folder on the computer you will be using.
- 2. Go to the ANSYS Customer Portal, https://support.ansys.com/training.

#### Note

If you do not have a login, you can request one by clicking **Customer Registration** on the log in page.

- 3. Enter the name of this tutorial into the search bar.
- 4. Narrow the results by using the filter on the left side of the page.
  - a. Click ANSYS Fluent under Product.
  - b. Click 18.0 under Version.
- 5. Select this tutorial from the list.
- 6. Click the **species\_transport\_R180.zip** link to download the input files.
- 7. Unzip species\_transport\_R180.zip to your working folder.

The file gascomb.msh can be found in the species\_transport folder created after unzipping the file.

8. Use Fluent Launcher to start the **2D** version of ANSYS Fluent.

Fluent Launcher displays your **Display Options** preferences from the previous session.

For more information about Fluent Launcher, see starting ANSYS Fluent using the Fluent Launcher in the Fluent Getting Started Guide.

- 9. Ensure that the **Display Mesh After Reading** and **Workbench Color Scheme** options are enabled.
- 10. Enable **Double-Precision**.
- 11. Ensure Serial is selected under Processing Options.

#### 13.5.2. Mesh

1. Read the mesh file gascomb.msh.

**File**  $\rightarrow$  Read  $\rightarrow$  Mesh...

After reading the mesh file, ANSYS Fluent will report that 1615 quadrilateral fluid cells have been read, along with a number of boundary faces with different zone identifiers.

### 13.5.3. General Settings

1. Check the mesh.

### **Setting Up Domain** $\rightarrow$ Mesh $\rightarrow$ Check

ANSYS Fluent will perform various checks on the mesh and will report the progress in the console. Ensure that the reported minimum volume reported is a positive number.

2. Scale the mesh.

Setting Up Domain  $\rightarrow$  Mesh  $\rightarrow$  Scale...

Since this mesh was created in units of millimeters, you will need to scale the mesh into meters.

💶 Scale M	esh			×
- Domain E	Extents			Scaling
Xmin (m)	0	Xmax (m)	1.8	Onvert Units
Ymin (m)	0	Ymax (m)	0.225	Specify Scaling Factors
				Mesh Was Created In
View Length Unit In m			mm     ▼       Scaling Factors       X     0.001       Y     0.001       Scale     Unscale	
Close Help				

- a. Select **mm** from the **Mesh Was Created In** drop-down list in the **Scaling** group box.
- b. Click Scale.
- c. Ensure that **m** is selected from the **View Length Unit In** drop-down list.
- d. Ensure that Xmax and Ymax are reset to 1.8 m and 0.225 m respectively.

The default SI units will be used in this tutorial, hence there is no need to change any units in this problem.

- e. Close the Scale Mesh dialog box.
- 3. Check the mesh.

Setting Up Domain  $\rightarrow$  Mesh  $\rightarrow$  Check

#### Note

You should check the mesh after you manipulate it (scale, convert to polyhedra, merge, separate, fuse, add zones, or smooth and swap). This will ensure that the quality of the mesh has not been compromised.

4. Examine the mesh with the default settings.

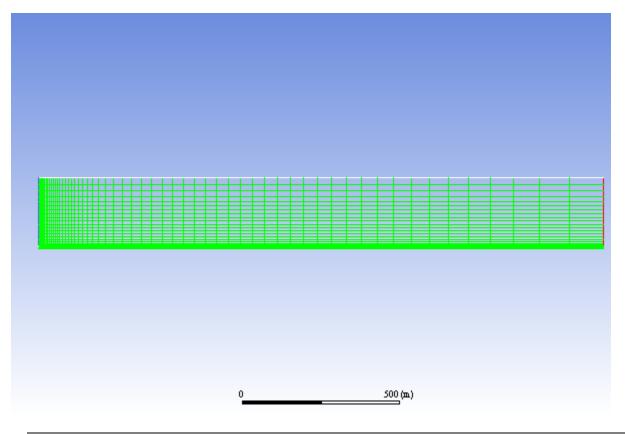
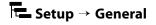


Figure 13.2: The Quadrilateral Mesh for the Combustor Model

#### Extra

You can use the right mouse button to probe for mesh information in the graphics window. If you click the right mouse button on any node in the mesh, information will be displayed in the ANSYS Fluent console about the associated zone, including the name of the zone. This feature is especially useful when you have several zones of the same type and you want to distinguish between them quickly.

5. Select **Axisymmetric** in the **2D Space** list.



General	
Mesh	
Scale	Check Report Quality
Display	
Solver	
Туре	Velocity Formulation
Pressure-Based	Absolute
Density-Based	Relative
Time	2D Space
Steady	Planar
Transient	Axisymmetric
	Axisymmetric Swirl
Gravity	Units
Help	

## 13.5.4. Models

1. Enable heat transfer by enabling the energy equation.

```
Setting Up Physics \rightarrow Models \rightarrow Energy
     Setting Up Physics
                           User Defined
                                           G Solving
                                                          Postprocessing
                                                                                 Viewing
                                                                                            Parallel
                                                                                                      Design
                                                                                                               ۵
 Models
Solver
Velocity Formulation
                   Operating Conditions...
                                                        Radiation...
                                                                                  Multiphase...
                                                                                                           🕰 Solidify/Melt...
 Absolute
                                             Energy Nat Heat Exchanger...
                                                                                  Species...
                     Reference Values...
                                                                                                           ID) Acoustics...
 Relative
                                                                                                          B More
                       Axisymmetric
                                                                                   Discrete Phase...
                                                Energy equation )US...
```

2. Select the standard k- $\varepsilon$  turbulence model.



Viscous Model		<b>—</b> ×	
Model	Model Constants		
Inviscid	Cmu		
<ul> <li>Laminar</li> <li>Spalart-Allmaras (1 eqn)</li> </ul>	0.09		
	C1-Epsilon		
k-epsilon (2 eqn)	1.44	=	
<ul> <li>k-omega (2 eqn)</li> <li>Transition k-kl-omega (3 eqn)</li> </ul>	C2-Epsilon		
<ul> <li>Transition SST (4 eqn)</li> </ul>	1.92		
Reynolds Stress (5 eqn)	TKE Prandtl Number		
$\odot$ Scale-Adaptive Simulation (SAS)	1		
Detached Eddy Simulation (DES)	TDR Prandtl Number		
k-epsilon Model	1.3	-	
Standard			
© RNG	User-Defined Functions		
Realizable	Turbulent Viscosity		
Near-Wall Treatment	none	•	
Standard Wall Functions	Prandtl Numbers		
Scalable Wall Functions	TKE Prandtl Number	<u> </u>	
<ul> <li>Non-Equilibrium Wall Functions</li> <li>Enhanced Wall Treatment</li> </ul>	none	-	
Menter-Lechner	TDR Prandtl Number		
<ul> <li>User-Defined Wall Functions</li> </ul>	none	<b>•</b>	
Ontinge	Energy Prandtl Number		
Options Options	none	<b>_</b>	
Production Kato-Launder	Wall Prandtl Number		
Production Limiter	none	• •	
OK Cancel Help			

a. Select k-epsilon (2 eqn) in the Model list.

The **Viscous Model** dialog box will expand to provide further options for the **k-epsilon** model.

- b. Retain the default settings for the **k-epsilon** model.
- c. Click **OK** to close the **Viscous Model** dialog box.
- 3. Enable chemical species transport and reaction.

**Setting Up Physics**  $\rightarrow$  Models  $\rightarrow$  Species...

Species Model	<b>X</b>	
Model Off Species Transport Non-Premixed Combustion Premixed Combustion Partially Premixed Combustion Composition PDF Transport	Mixture Properties Mixture Material methane-air View Import CHEMKIN Mechanism Number of Volumetric Species 5	
Reactions Volumetric Wall Surface Particle Surface Electrochemical Chemistry Solver None - Explicit Source	Turbulence-Chemistry Interaction Finite-Rate/No TCI Finite-Rate/Eddy-Dissipation Eddy-Dissipation Coal Calculator	
Options Inlet Diffusion Diffusion Energy Source Full Multicomponent Diffusion Thermal Diffusion OK Apply	Select Boundary Species Select Monitored Species	

a. Select Species Transport in the Model list.

The **Species Model** dialog box will expand to provide further options for the **Species Transport** model.

- b. Enable **Volumetric** in the **Reactions** group box.
- c. Select methane-air from the Mixture Material drop-down list.

Scroll down the list to find **methane-air**.

#### Note

The **Mixture Material** list contains the set of chemical mixtures that exist in the ANSYS Fluent database. You can select one of the predefined mixtures to access a complete description of the reacting system. The chemical species in the system and their physical and thermodynamic properties are defined by your selection of the mixture material. You can alter the mixture material selection or modify the mixture material properties using the **Create/Edit Materials** dialog box (see Materials (p. 545)).

d. Select Eddy-Dissipation in the Turbulence-Chemistry Interaction group box.

The eddy-dissipation model computes the rate of reaction under the assumption that chemical kinetics are fast compared to the rate at which reactants are mixed by turbulent fluctuations (eddies).

e. Click OK to close the Species Model dialog box.

Prior to listing the properties that are required for the models you have enabled, ANSYS Fluent will display a warning about the symmetry zone in the console. You may have to scroll up to see this warning.

Warning: It appears that symmetry zone 5 should actually be an axis (it has faces with zero area projections). Unless you change the zone type from symmetry to axis, you may not be able to continue the solution without encountering floating point errors.

In the axisymmetric model, the boundary conditions should be such that the centerline is an axis type instead of a symmetry type. You will change the symmetry zone to an axis boundary in Boundary Conditions (p. 548).

#### 13.5.5. Materials

In this step, you will examine the default settings for the mixture material. This tutorial uses mixture properties copied from the Fluent Database. In general, you can modify these or create your own mixture properties for your specific problem as necessary.

1. Confirm the properties for the mixture materials.

# **F**Setup $\rightarrow$ Materials $\rightarrow$ Mixture $\rightarrow$ methane-air $\stackrel{\bigcirc}{\Box}$ Edit...

The **Create/Edit Materials** dialog box will display the mixture material (**methane-air**) that was selected in the **Species Model** dialog box. The properties for this mixture material have been copied from the **Fluent Database...** and will be modified in the following steps.

Create/Edit Materials				
Name	Material Type	Order Materials by		
methane-air	mixture	<ul> <li>Name</li> </ul>		
Chemical Formula	Fluent Mixture Materials	Chemical Formula		
	methane-air	<ul> <li>Fluent Database</li> </ul>		
	Mixture			
	none	User-Defined Database		
Properties				
Mixture Species names   Edit				
Reaction eddy-dissipation				
Mechanism reaction-mechs	▼ Edit			
Density (kg/m3) incompressible-ideal	-gas			
Change/Create Delete Close Help				

a. Click the **Edit...** button to the right of the **Mixture Species** drop-down list to open the **Species** dialog box.

Species	
Mixture methane-air	
Available Materials	Selected Species
air	ch4 o2 co2 h2o n2 Add Remove
Selected Site Species	Selected Solid Species
Add Remove	Add Remove
	OK Cancel Help

You can add or remove species from the mixture material as necessary using the **Species** dialog box.

i. Retain the default selections from the **Selected Species** selection list.

The species that make up the methane-air mixture are predefined and require no modification.

- ii. Click **OK** to close the **Species** dialog box.
- b. Click the Edit... button to the right of the Reaction drop-down list to open the Reactions dialog box.

lixture methane-air	•		Total Nu	mber of Reactions	1 🜩
Reaction Name reaction-1		action Type Volumetric 🔿 \	Vall Surface 🔘 Part	icle Surface 🔘 Elec	trochemical
Number of Reactan	ts 2 🌩		Number of Prod	ucts 2 🖨	
Species	Stoich. Coefficient	Rate Exponent	Species	Stoich. Coefficient	Rate Exponent
ch4	▼ 1	1	E Co2	▼ 1	0
02	▼ 2	1	h2o	▼ 2	0
Arrhenius Rate			Mixing Rate		
Pre-Exponent	tial Factor 2.119	e+11	A 4	B 0.5	
Activation Energy	(j/kgmol) 2.027	e+08			
Temperature B	Exponent 0				
🗌 Include Backwa	ard Reaction	Specify			
Third-Body Effi	ciencies	Specify			
Pressure-Deper	ndent Reaction	Specify			
Coverage-Depe	endent Reaction	Specify			

The eddy-dissipation reaction model ignores chemical kinetics (the Arrhenius rate) and uses only the parameters in the **Mixing Rate** group box in the **Reactions** dialog box. The **Arrhenius Rate** group box will therefore be inactive. The values for **Rate Exponent** and **Arrhenius Rate** parameters are included in the database and are employed when the alternate finite-rate/eddy-dissipation model is used.

- i. Retain the default values in the Mixing Rate group box.
- ii. Click **OK** to close the **Reactions** dialog box.
- c. Retain the selection of incompressible-ideal-gas from the Density drop-down list.
- d. Retain the selection of **mixing-law** from the **Cp** (**Specific Heat**) drop-down list.
- e. Retain the default values for Thermal Conductivity, Viscosity, and Mass Diffusivity.

Create/Edit Materials						×
Name		Material Type			Order Materials by	
methane-air		mixture			Name	
Chemical Formula		Fluent Mixture Mate	rials		Chemical Formula	
		methane-air			Fluent Database.	
		Mixture				
		none			User-Defined Databa	se
Properties						
Cp (Specific Heat) (j/kg-k)	mixing-law		▼ Edit	*		
Thermal Conductivity (w/m-k)			• Edit			
	0.0454					
Viscosity (kg/m-s)	constant		<ul> <li>Edit</li> </ul>			
	1.72e-05			=		
Mass Diffusivity (m2/s)	constant-dilute-appx	<	• Edit			
	2.88e-05			-		
	Chan	ge/Create Delete	Close Help	Þ		

- f. Click Change/Create to accept the material property settings.
- g. Close the Create/Edit Materials dialog box.

The calculation will be performed assuming that all properties except density and specific heat are constant. The use of constant transport properties (viscosity, thermal conductivity, and mass diffusivity coefficients) is acceptable because the flow is fully turbulent. The molecular transport properties will play a minor role compared to turbulent transport.

## 13.5.6. Boundary Conditions

Setup → ↔ Boundary Conditions

Boundary Conditions
Zone Filter Text
interior-4
pressure-outlet-9
symmetry-5
velocity-inlet-6
velocity-inlet-8
wall-2 wall-7
waii-7
Phase Type ID
mixture v symmetry v 5
Edit Copy Profiles
Parameters Operating Conditions
Display Mesh
Periodic Conditions
Help
( ) Cip

1. Convert the symmetry zone to the axis type.

# **Setup** $\rightarrow$ **Conditions** $\rightarrow \stackrel{\frown}{=}$ symmetry-5

The symmetry zone must be converted to an axis to prevent numerical difficulties where the radius reduces to zero.

- a. Select **axis** from the **Type** drop-down list.
- 2. Set the boundary conditions for the air inlet (velocity-inlet-8).



To determine the zone for the air inlet, display the mesh without the fluid zone to see the boundaries. Use the right mouse button to probe the air inlet. ANSYS Fluent will report the zone name (**velocity-inlet-8**) in the console.

Velocity Inlet								×
Zone Name								
air-inlet								
Momentum	Thermal	Radiation	Species	DPM	Mul	tiphase	Potential	UDS
Velocit	y Specificatio	on Method M	agnitude, N	ormal to	Bound	lary		•
	Refere	nce Frame	bsolute					•
	Velocity Ma	gnitude (m/s	) 0.5			cons	tant	•
Supersonic/Init	ial Gauge Pre	essure (pascal	) 0			cons	tant	•
	- Turbulence	e						
	Specificatio	n Method Int	tensity and I	Hydraulic	Diame	ter		-
			Turbulent	Intensity	(%)	10		P
			Hydraulic	Diameter	(m)	.44		P
		0	K Cancel	Help				

a. Enter air-inlet for Zone Name.

This name is more descriptive for the zone than velocity-inlet-8.

- b. Enter 0.5 m/s for **Velocity Magnitude**.
- c. Select **Intensity and Hydraulic Diameter** from the **Specification Method** drop-down list in the **Tur-bulence** group box.
- d. Enter 10 % for **Turbulent Intensity**.
- e. Enter 0.44 *m* for **Hydraulic Diameter**.
- f. Click the **Thermal** tab and retain the default value of 300 *K* for **Temperature**.
- g. Click the Species tab and enter 0.23 for o2 in the Species Mass Fractions group box.

💶 Velocity	Inlet					×
Zone Name	9					
air-inlet						
Momentu	um Thermal	Radiation Species	DPM	Multiphase	Potential	UDS
	Specify Spec	ies in Mole Fractions				
Species M	ass Fractions					
ch4	0	constant		•		
o2	0.23	constant		•		
co2	0	constant		•		
h2o	0	constant		•		
		OK Canc	Help			

- h. Click **OK** to close the **Velocity Inlet** dialog box.
- 3. Set the boundary conditions for the fuel inlet (velocity-inlet-6).

Setup $\rightarrow$ Boundary Conditions $\rightarrow$ velocity-inlet-6	₫́ Edit
--	---------

💶 Velocity Inlet								×
Zone Name								
fuel-inlet								
Momentum	Thermal	Radiation	Species	DPM	Mul	tiphase	Potential	UDS
Velocit	y Specificatio	on Method M	agnitude, N	ormal to	Bound	dary		•
	Refere	nce Frame A	bsolute					•
	Velocity Ma	gnitude (m/s	) 80			cons	tant	•
Supersonic/Init	ial Gauge Pre	essure (pascal	) 0			cons	tant	•
	- Turbulence	9						
	Specificatio	n Method Int	tensity and I	Hydraulic	Diame	eter		•
			Turbulent	Intensity	(%)	10		P
			Hydraulic	Diameter	(m)	0.01		P
		0	K Cancel	Help				łł.

a. Enter fuel-inlet for Zone Name.

This name is more descriptive for the zone than **velocity-inlet-6**.

- b. Enter 80 m/s for the **Velocity Magnitude**.
- c. Select **Intensity and Hydraulic Diameter** from the **Specification Method** drop-down list in the **Turbulence** group box.
- d. Enter 10 % for **Turbulent Intensity**.
- e. Enter 0.01 *m* for **Hydraulic Diameter**.
- f. Click the **Thermal** tab and retain the default value of 300 *K* for **Temperature**.
- g. Click the **Species** tab and enter 1 for **ch4** in the **Species Mass Fractions** group box.
- h. Click OK to close the Velocity Inlet dialog box.
- 4. Set the boundary conditions for the exit boundary (pressure-outlet-9).

**Example :** Setup  $\rightarrow$  Boundary Conditions  $\rightarrow$  pressure-outlet-9  $\stackrel{\bigcirc}{\rightarrow}$  Edit...

Pressure Outlet
Zone Name
pressure-outlet-9
Momentum Thermal Radiation Species DPM Multiphase Potential UDS
Backflow Reference Frame Absolute
Gauge Pressure (pascal) 0 constant
Backflow Direction Specification Method Normal to Boundary
Backflow Pressure Specification Total Pressure
Average Pressure Specification
Target Mass Flow Rate
Turbulence
Specification Method Intensity and Hydraulic Diameter
Backflow Turbulent Intensity (%) 10
Backflow Hydraulic Diameter (m) 0.45
OK Cancel Help

- a. Retain the default value of 0 *Pa* for **Gauge Pressure**.
- b. Select Intensity and Hydraulic Diameter from the Specification Method drop-down list in the Turbulence group box.
- c. Enter 10 % for **Backflow Turbulent Intensity**.
- d. Enter 0.45 *m* for **Backflow Hydraulic Diameter**.

- e. Click the **Thermal** tab and retain the default value of 300 *K* for **Backflow Total Temperature**.
- f. Click the Species tab and enter 0.23 for o2 in the Species Mass Fractions group box.
- g. Click **OK** to close the **Pressure Outlet** dialog box.

The **Backflow** values in the **Pressure Outlet** dialog box are utilized only when backflow occurs at the pressure outlet. Always assign reasonable values because backflow may occur during intermediate iterations and could affect the solution stability.

5. Set the boundary conditions for the outer wall (**wall-7**).

# **E** Setup $\rightarrow$ Boundary Conditions $\rightarrow$ wall-7 $\stackrel{\text{D}}{\hookrightarrow}$ Edit...

Use the mouse-probe method described for the air inlet to determine the zone corresponding to the outer wall.

💶 Wall		×
Zone Name outer-wall Adjacent Cell Zone fluid-1		
Momentum Thermal	Radiation Species DPM Multiphase UDS Wall Film Potential	
Thermal Conditions <ul> <li>Heat Flux</li> <li>Temperature</li> <li>Convection</li> <li>Radiation</li> <li>Mixed</li> <li>via System Coupling</li> <li>via Mapped Interface</li> </ul> Material Name   aluminum	Temperature (k) 300 constant Wall Thickness (m) 0 Heat Generation Rate (w/m3) 0 constant	T
	OK Cancel Help	.1

a. Enter outer-wall for Zone Name.

This name is more descriptive for the zone than wall-7.

- b. Click the **Thermal** tab.
  - i. Select Temperature in the Thermal Conditions list.
  - ii. Retain the default value of 300 *K* for **Temperature**.
- c. Click **OK** to close the **Wall** dialog box.

6. Set the boundary conditions for the fuel inlet nozzle (wall-2).

**E** Setup  $\rightarrow$  Boundary Conditions  $\rightarrow$  wall-2  $\stackrel{\square}{\hookrightarrow}$  Edit...

🖬 Wall		×
Zone Name nozzle Adjacent Cell Zone fluid-1		
Momentum Thermal	Radiation Species DPM Multiphase UDS Wall Film Potential	
Thermal Conditions <ul> <li>Heat Flux</li> <li>Temperature</li> <li>Convection</li> <li>Radiation</li> <li>Mixed</li> <li>via System Coupling</li> <li>via Mapped Interface</li> </ul> Material Name   aluminum	Heat Flux (w/m2) 0 constant Wall Thickness (m) 0 Heat Generation Rate (w/m3) 0 constant	▼ ₽ ▼
	OK Cancel Help	.1

a. Enter nozzle for **Zone Name**.

This name is more descriptive for the zone than wall-2.

- b. Click the **Thermal** tab.
  - i. Retain the default selection of Heat Flux in the Thermal Conditions list.
  - ii. Retain the default value of 0  $W / m^2$  for **Heat Flux**, so that the wall is adiabatic.
- c. Click OK to close the Wall dialog box.

#### **13.5.7. Initial Reaction Solution**

You will first calculate a solution for the basic reacting flow neglecting pollutant formation. In a later step, you will perform an additional analysis to simulate NOx.

1. Select the Coupled Pseudo Transient solution method.



Solution Methods	
Pressure-Velocity Coupling	
Scheme	
Coupled 👻	
Spatial Discretization	_
Gradient	
Least Squares Cell Based 🔻	
Pressure	=
Second Order 🔻	-
Momentum	
Second Order Upwind 🔻	
Turbulent Kinetic Energy	
First Order Upwind 🔻	
Turbulent Dissipation Rate	
First Order Upwind 🔻	÷
Transient Formulation	
Non-Iterative Time Advancement	
Frozen Flux Formulation	
Pseudo Transient	
Warped-Face Gradient Correction	
High Order Term Relaxation Options	
Set All Species Discretizations Together	
Default	
Help	

- a. Select **Coupled** from the **Scheme** drop-down list in the **Pressure-Velocity Coupling** group box.
- b. Retain the default selections in the **Spatial Discretization** group box.
- c. Enable Pseudo Transient.

The Pseudo Transient option enables the pseudo transient algorithm in the coupled pressure-based solver. This algorithm effectively adds an unsteady term to the solution equations in order to improve stability and convergence behavior. Use of this option is recommended for general fluid flow problems.

2. Modify the solution controls.

**Solving**  $\rightarrow$  Controls  $\rightarrow$  Controls...

Solution Controls	
Pseudo Transient Explicit Relaxation Factors	
Pressure	1
0.5	
Momentum	Ξ
0.5	
Density	۳
0.25	
Body Forces	
1	
Turbulent Kinetic Energy	
0.75	
Turkulant Dissignting Data	Ŧ
Default	
Equations Limits Advanced	
Set All Species URFs Together	

a. Enter 0.25 under **Density** in the **Pseudo Transient Explicit Relaxation Factors** group box.

The default explicit relaxation parameters in ANSYS Fluent are appropriate for a wide range of general fluid flow problems. However, in some cases it may be necessary to reduce the relaxation factors to stabilize the solution. Some experimentation is typically necessary to establish the optimal values. For this tutorial, it is sufficient to reduce the density explicit relaxation factor to 0.25 for stability.

b. Click Advanced... to open the Advanced Solution Controls dialog box and select the Expert tab.

The Expert tab in the Advanced Solution Controls dialog box allows you to individually specify the solution method and Pseudo Transient Time Scale Factors for each equation, except for the flow equations. When using the Pseudo Transient method for general reacting flow cases, increasing the species and energy time scales is recommended.

Advanced Solution Controls			×	
Multigrid	Multi-Stage	Expert		
Spatial Discretization Limiter Limiter Type Standard				
<ul> <li>Cell to Face Limiting</li> <li>Cell to Cell Limiting</li> <li>Apply Limiter Filter</li> </ul>				
Pseudo Transient Method Usage				
	On/Off Under-Rela	axation Factor Time Scale Fac	ctor	
Turbulent Kinetic Energy	.8	1		
Turbulent Dissipation Rate	.8	1		
ch4	☑ 1	10		
o2	☑ 1	10		
co2	☑ 1	10		
h2o	☑ 1	10		
Energy	☑ 1	10		
Default				
OK Cancel Help				

- i. Enable the pseudo-transient method for **ch4**, **o2**, **co2**, **h2o**, and **Energy** in the **Expert** tab, by selecting each one under **On/Off**.
- ii. Enter 10 for the Time Scale Factor for ch4, o2, co2, h2o, and Energy.
- iii. Click OK to close the Advanced Solution Controls dialog box.
- 3. Ensure the plotting of residuals during the calculation.

Solving  $\rightarrow$  Reports  $\rightarrow$  Residuals...

Residual Monitors					×	
Options	Equations					
Print to Console	Residual	Monito	r Check Converge	ence Absolute Criteria		
V Plot	continuity	<b>V</b>		0.001	E	
Window	x-velocity	<b>V</b>	<b>V</b>	0.001		
1 Curves Axes	y-velocity	<b>V</b>	$\checkmark$	0.001		
Iterations to Plot	energy	<b>V</b>	$\checkmark$	1e-06		
1000	1	[ma]	Ima	0.004		
1000	Residual Values Convergence Criterion				n	
	Normalize		Iterations	absolute	•	
Iterations to Store			5			
1000 🚖	Scale			Convergence Condi	tions	
Compute Local Scale						
OK Plot Renormalize Cancel Help						

- a. Ensure that **Plot** is enabled in the **Options** group box.
- b. Click **OK** to close the **Residual Monitors** dialog box.
- 4. Initialize the field variables.

Solving → Initialization

Initialization								
Method		Patch						
O Hybrid	More Settings	Reset Statistics						
Standard	Options	Reset DPM	t = 0 Initialize					

- a. Retain the default **Hybrid** initialization method and click **Initialize** to initialize the variables.
- 5. Save the case file (gascombl.cas.gz).

```
File \rightarrow Write \rightarrow Case...
```

- a. Enter gascomb1.cas.gz for Case File.
- b. Ensure that Write Binary Files is enabled to produce a smaller, unformatted binary file.
- c. Click OK to close the Select File dialog box.
- 6. Run the calculation by requesting 200 iterations.

```
Solving \rightarrow Run Calculation \rightarrow Advanced...
```

Run Calculation	
Check Case	Update Dynamic Mesh
Pseudo Transient Option Fluid Time Scale	ons
<ul> <li>Time Step Method</li> <li>User Specified</li> <li>Automatic</li> </ul>	Timescale Factor 5
Length Scale Method	Verbosity
Aggressive 🔹	0
Aggressive	
Number of Iterations 200 🐳 Profile Update Interval	Reporting Interval
Number of Iterations	Reporting Interval
Number of Iterations 200 🗣 Profile Update Interval	Reporting Interval
Number of Iterations 200 🗣 Profile Update Interval 1 🗣	Reporting Interval

a. Select Aggressive from the Length Scale Method drop-down list.

When using the Automatic Time Step Method ANSYS Fluent computes the Pseudo Transient time step based on characteristic length and velocity scales of the problem. The Conservative Length Scale Method uses the smaller of two computed length scales emphasizing solution stability. The Aggressive Length Scale Method uses the larger of the two which may provide faster convergence in some cases.

b. Enter 5 for the Timescale Factor.

The Timescale Factor allows you to further manipulate the computed Time Step calculated by ANSYS Fluent. Larger time steps can lead to faster convergence. However, if the time step is too large it can lead to solution instability.

- c. Enter 200 for Number of Iterations.
- d. Click Calculate.

The solution will converge after approximately 160 iterations.

7. Save the case and data files (gascombl.cas.gz and gascombl.dat.gz).

### File $\rightarrow$ Write $\rightarrow$ Case & Data...

#### Note

If you choose a file name that already exists in the current folder, ANSYS Fluent will ask you to confirm that the previous file is to be overwritten.

### 13.5.8. Postprocessing

*Review the solution by examining graphical displays of the results and performing surface integrations at the combustor exit.* 

1. Report the total sensible heat flux.

Flux Reports Options	Boundaries Filter Text	Pacute
<ul> <li>Mass Flow Rate</li> <li>Total Heat Transfer Rate</li> <li>Total Sensible Heat Transfer Rate</li> <li>Radiation Heat Transfer Rate</li> </ul>	air-inlet axis-5 fuel-inlet interior-4 nozzle outer-wall pressure-outlet-9	173.7963996081019 16.64988560719532 -0 -11787.92355890429 -192029.9785504543
	٠	٠
Save Output Parameter		Heat of Reaction Source (w)
		203633.3
		Net Results (w)
		5.823312
	Compute Write Close Help	ł.

Postprocessing  $\rightarrow$  Reports  $\rightarrow$  Fluxes...

- a. Select Total Sensible Heat Transfer Rate in the Options list.
- b. Select all the boundaries from the **Boundaries** selection list (you can click the select-all button (1975)).
- c. Click Compute and close the Flux Reports dialog box.

### Note

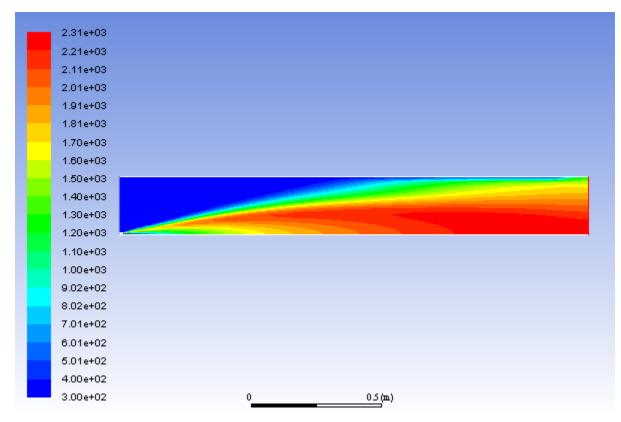
The energy balance is good because the net result is small compared to the heat of reaction.

2. Display filled contours of temperature (Figure 13.3: Contours of Temperature (p. 561)).

Postprocessing  $\rightarrow$  Graphics  $\rightarrow$  Contours  $\rightarrow$  Edit...

- a. Ensure that **Filled** is enabled in the **Options** group box.
- b. Select Temperature... and Static Temperature in the Contours of drop-down lists.
- c. Click Display.





The peak temperature is approximately 2310 K.

3. Display velocity vectors (Figure 13.4: Velocity Vectors (p. 563)).

Postprocessing  $\rightarrow$  Graphics  $\rightarrow$  Vectors  $\rightarrow$  Edit...

Vectors	
Options Global Range Auto Range Clip to Range Auto Scale Draw Mesh	Vectors of Velocity
Style arrow	Min (m/s) Max (m/s) 0.4037463 82.27368
Scale Skip 0.01 0	Surfaces Filter Text
Vector Options Custom Vectors	axis-5 fuel-inlet interior-4 nozzle outer-wall
	Display Compute Close Help

- a. Enter 0.01 for Scale.
- b. Click the Vector Options... button to open the Vector Options dialog box.

Vector Options	×
🔲 In Plane	Scale Head
🗹 Fixed Length	0.1
🗹 X Component	
Y Component	
Z Component	
	Color
Apply	Close Help

i. Enable Fixed Length.

The fixed length option is useful when the vector magnitude varies dramatically. With fixed length vectors, the velocity magnitude is described only by color instead of by both vector length and color.

- ii. Click Apply and close the Vector Options dialog box.
- c. Click **Display** and close the **Vectors** dialog box.

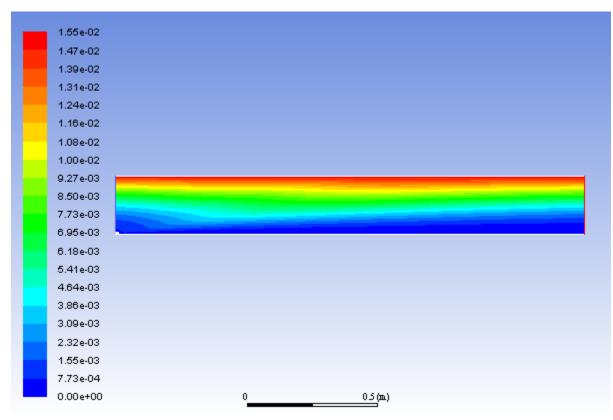
### Figure 13.4: Velocity Vectors

8.23e+01	
7.82e+01	
7.41e+01	
7.00e+01	
6.59e+01	
6.18e+01	
5.77e+01	
5.36e+01	
4.95e+01	
4.54e+01	
4.13e+01	
3.72e+01	
3.32e+01	
2.91e+01	
2.50e+01	
2.09e+01	
1.68e+01	
1.27e+01	
8.59e+00	
4.04e-01	0 0.5(m.)

4. Display filled contours of stream function (Figure 13.5: Contours of Stream Function (p. 564)).

Postprocessing  $\rightarrow$  Graphics  $\rightarrow$  Contours  $\rightarrow$  Edit...

- a. Select Velocity... and Stream Function from the Contours of drop-down lists.
- b. Click **Display**.



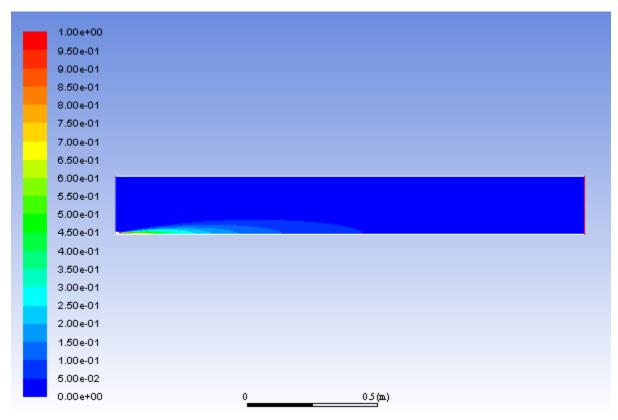
#### Figure 13.5: Contours of Stream Function

The entrainment of air into the high-velocity methane jet is clearly visible in the streamline display.

5. Display filled contours of mass fraction for CH<sub>4</sub> (Figure 13.6: Contours of CH4 Mass Fraction (p. 565)).

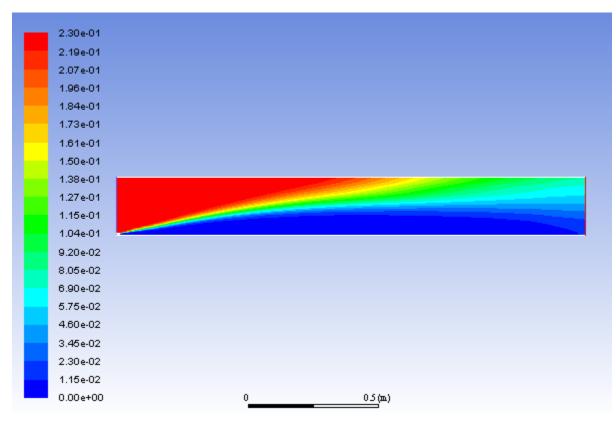
Postprocessing  $\rightarrow$  Graphics  $\rightarrow$  Contours  $\rightarrow$  Edit...

- a. Select Species... and Mass fraction of ch4 from the Contours of drop-down lists.
- b. Click **Display**.



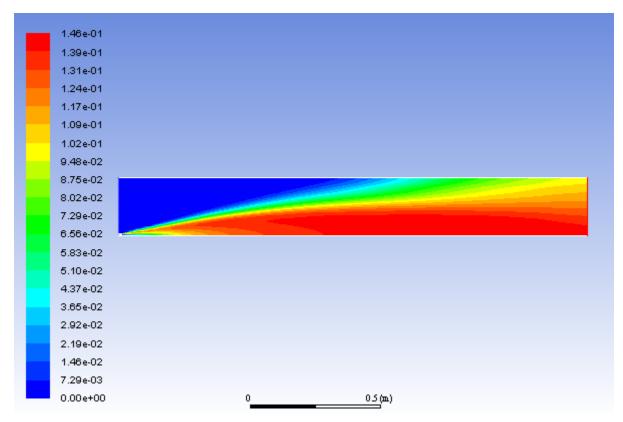
### Figure 13.6: Contours of CH4 Mass Fraction

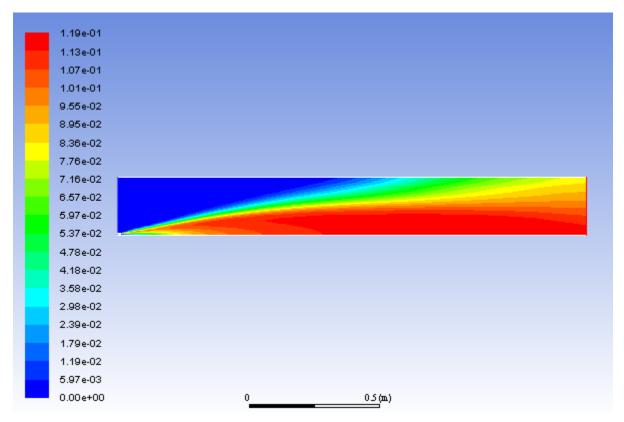
6. In a similar manner, display the contours of mass fraction for the remaining species O<sub>2</sub>, CO<sub>2</sub>, and H<sub>2</sub>O (Figure 13.7: Contours of O2 Mass Fraction (p. 566), Figure 13.8: Contours of CO2 Mass Fraction (p. 566), and Figure 13.9: Contours of H2O Mass Fraction (p. 567)) Close the **Contours** dialog box when all of the species have been displayed.



#### Figure 13.7: Contours of O2 Mass Fraction

#### Figure 13.8: Contours of CO2 Mass Fraction





### Figure 13.9: Contours of H2O Mass Fraction

7. Determine the average exit temperature.

Postprocessing  $\rightarrow$  Reports  $\rightarrow$  Surface Integrals...

Surface Integrals	
Report Type	Field Variable
Mass-Weighted Average	Temperature 🔻
Custom Vectors	Static Temperature 👻
Vectors of	Surfaces Filter Text
Custom Vectors	air-inlet
	axis-5
Save Output Parameter	fuel-inlet
	interior-4
	nozzle
	outer-wall
	pressure-outlet-9
	Mass-Weighted Average (k)
	1839.818
Compute	rite Close Help

- a. Select Mass-Weighted Average from the Report Type drop-down list.
- b. Select Temperature... and Static Temperature from the Field Variable drop-down lists.

The mass-averaged temperature will be computed as:

$$\overline{T} = \frac{\int T\rho \vec{v} \cdot d\vec{A}}{\int \rho \vec{v} \cdot d\vec{A}}$$
(13.2)

- c. Select **pressure-outlet-9** from the **Surfaces** selection list, so that the integration is performed over this surface.
- d. Click Compute.

The **Mass-Weighted Average** field will show that the exit temperature is approximately 1840 K.

8. Determine the average exit velocity.

Postprocessing  $\rightarrow$  Reports  $\rightarrow$  Surface Integrals...

Surface Integrals	
Report Type	Field Variable
Area-Weighted Average 🔹	Velocity 🔻
Custom Vectors	Velocity Magnitude 🗸 🗸
Vectors of	Surfaces Filter Text
Custom Vectors	air-inlet
	axis-5
Save Output Parameter	fuel-inlet
	interior-4
	nozzle
	outer-wall
	pressure-outlet-9
	Area-Weighted Average (m/s)
	3.305419
Compute W	/rite Close Help

- a. Select Area-Weighted Average from the Report Type drop-down list.
- b. Select Velocity... and Velocity Magnitude from the Field Variable drop-down lists.

The area-weighted velocity-magnitude average will be computed as:

$$\overline{v} = \frac{1}{A} \int v dA \tag{13.3}$$

c. Click Compute.

The **Area-Weighted Average** field will show that the exit velocity is approximately 3.30 m/s.

d. Close the Surface Integrals dialog box.

## 13.5.9. NOx Prediction

In this section you will extend the ANSYS Fluent model to include the prediction of NOx. You will first calculate the formation of both thermal and prompt NOx, then calculate each separately to determine the contribution of each mechanism.

11

1. Enable the NOx model.

**Setup** 
$$\rightarrow$$
 Models  $\rightarrow$  Species  $\rightarrow$  NOx  $\stackrel{\bigcirc}{\rightarrow}$  Edit...

NOx Model	
Models	Formation Model Parameters
Formation Reduction Turbulence Interaction Mode	Thermal Prompt Fuel N2O Path
Pathways  Thermal NOx  Prompt NOx  Fuel NOx  N20 Intermediate  Fuel Streams  Fuel Streams  Fuel Stream ID  Fuel Species [1/5]  Ch4  o2  co2  h2o  User-Defined Functions  Place Fuel Stream	[O] Model partial-equilibrium   [OH] Model none
NOx Rate none 🔻	
Apply Close H	Help

- a. Enable Thermal NOx and Prompt NOx in the Pathways group box.
- b. Select **ch4** from the **Fuel Species** selection list.
- c. Click the Turbulence Interaction Mode tab.

NOx Model	<b>—</b>
Models	Formation Model Parameters
Formation Reduction Turbulence Interaction Mode	Thermal Prompt Fuel N2O Path
PDF Mode temperature  PDF Type beta PDF Points 20 Temperature Variance transported Tmax Option global-tmax	[O] Model partial-equilibrium   [OH] Model none
Apply Close H	Help

i. Select temperature from the PDF Mode drop-down list.

This will enable the turbulence-chemistry interaction. If turbulence interaction is not enabled, you will be computing NOx formation without considering the important influence of turbulent fluctuations on the time-averaged reaction rates.

ii. Retain the default selection of **beta** from the **PDF Type** drop-down list and enter 20 for **PDF Points**.

The value for **PDF Points** is increased from 10 to 20 to obtain a more accurate NOx prediction.

- iii. Select transported from the Temperature Variance drop-down list.
- d. Select **partial-equilibrium** from the **[O] Model** drop-down list in the **Formation Model Parameters** group box in the **Thermal** tab.

The partial-equilibrium model is used to predict the O radical concentration required for thermal NOx prediction.

e. Click the **Prompt** tab.

NOx Model	×
Models	Formation Model Parameters
Formation Reduction Turbulence Interaction Mode	Thermal Prompt Fuel N2O Path
PDF Mode temperature   PDF Type beta	Fuel Carbon Number 1 Equivalence Ratio 0.76
PDF Points 20	
Apply Close	Help

- i. Retain the default value of 1 for **Fuel Carbon Number**.
- ii. Enter 0.76 for Equivalence Ratio.

All of the parameters in the **Prompt** tab are used in the calculation of prompt NOx formation. The **Fuel Carbon Number** is the number of carbon atoms per molecule of fuel. The **Equivalence Ratio** defines the fuel-air ratio (relative to stoichiometric conditions).

- f. Click **Apply** to accept these changes and close the **NOx Model** dialog box.
- 2. Enable the calculation of NO species only and temperature variance.



Equations
Equations [2/9]
Flow
Turbulence
ch4
o2
co2
h2o
Pollutant no
Temperature Variance
Energy
OK Default Cancel Help

- a. Deselect all variables except **Pollutant no** and **Temperature Variance** from the **Equations** selection list.
- b. Click **OK** to close the **Equations** dialog box.

You will predict NOx formation in a "postprocessing" mode, with the flow field, temperature, and hydrocarbon combustion species concentrations fixed. Hence, only the NO equation will be computed. Prediction of NO in this mode is justified on the grounds that the NO concentrations are very low and have negligible impact on the hydrocarbon combustion prediction.

3. Modify the solution controls for **Pollutant no** and **Temperature Variance**.

Solving $\rightarrow$ Controls $\rightarrow$ Controls
---

Solution Controls	
Pseudo Transient Explicit Relaxation Factors	_
0.75	^
h2o	
0.75	
Pollutant no	
1	
Temperature Variance	
1	
Energy	Ξ
0.75	
	-
Default	
Equations Limits Advanced	
Set All Species URFs Together	

- a. Click Advanced... to open the Advanced Solution Controls dialog box.
  - i. In the **Expert** tab, enable the pseudo-transient method for **Pollutant no** and **Temperature Variance**, by selecting them under **On/Off**.
  - ii. Enter 10 for Time Scale Factor for Pollutant no and Temperature Variance.
  - iii. Click OK to close the Advanced Solution Controls dialog box.
- b. Enter 1 for **Pollutant no** and **Temperature Variance** in the **Pseudo Transient Explicit Relaxation Factors** group box.
- 4. Confirm the convergence criterion for the NO species equation.

So	olving $\rightarrow$	$\textbf{Reports} \rightarrow$	Residuals
----	----------------------	--------------------------------	-----------

Residual Monitors					×
Options	Equations				
Print to Console	Residual	Monito	r Check Converg	ence Absolute Criteria	
V Plot	pollut_no		$\checkmark$	1e-06	
Window	tvar	V	$\checkmark$	0.001	
1   Curves   Axes     Iterations to Plot   1000     1000   Iterations to Store     1000   Iterations to Store	Residual Values Normalize Scale Compute Loca	l Scale	Iterations	Convergence Criterion absolute Convergence Conditio	• ns
OK	Plot Renormali	ce Ca	ncel Help		

- a. Ensure that the **Absolute Criteria** for **pollut\_no** is set to 1e-06.
- b. Click OK to close the Residual Monitors dialog box.
- 5. Request 25 more iterations.

### **E**Solution $\rightarrow$ Run Calculation

The solution will converge in approximately 10 iterations.

6. Save the new case and data files (gascomb2.cas.gz and gascomb2.dat.gz).

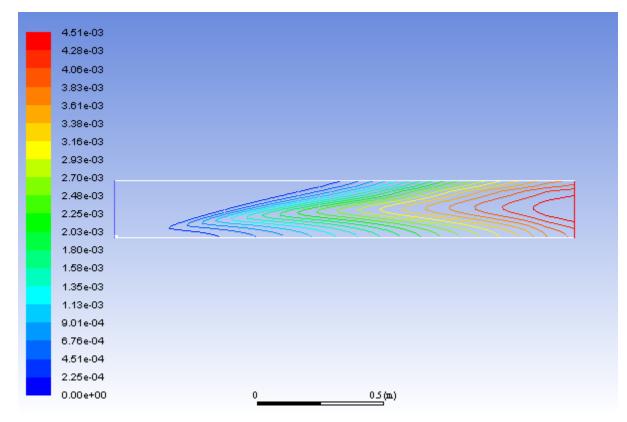
100001 10001 10	File $\rightarrow$	Write →	Case	&	Data
-----------------	--------------------	---------	------	---	------

7. Review the solution by creating and displaying a contour definition for NO mass fraction (Figure 13.10: Contours of NO Mass Fraction — Prompt and Thermal NOx Formation (p. 575)).

Postprocessing  $\rightarrow$  Graphics  $\rightarrow$  Contours  $\rightarrow$  New...

- a. Enter contour-no-mass-fraction for Contour Name.
- b. Disable Filled in the Options group box.
- c. Select NOx... and Mass fraction of Pollutant no from the Contours of drop-down lists.
- d. Click Save/Display and close the Contours dialog box.

Figure 13.10: Contours of NO Mass Fraction — Prompt and Thermal NOx Formation



8. Calculate the average exit NO mass fraction.

Postprocessing  $\rightarrow$  Reports  $\rightarrow$  Surface Integrals...

Surface Integrals	
Report Type	Field Variable
Mass-Weighted Average	N0x 🔻
Custom Vectors	Mass fraction of Pollutant no
Vectors of	Surfaces Filter Text
Custom Vectors	air-inlet
	axis-5
Save Output Parameter	fuel-inlet
	interior-4
	nozzle
	outer-wall
	pressure-outlet-9 zone-surface-7
	zone-sunace-/
	Mass-Weighted Average
	0.0042108
Compute	rite Close Help

- a. Select Mass-Weighted Average from the Report Type drop-down list.
- b. Select NOx... and Mass fraction of Pollutant no from the Field Variable drop-down lists.
- c. Ensure that **pressure-outlet-9** is selected from the **Surfaces** selection list.
- d. Click Compute.

The **Mass-Weighted Average** field will show that the exit NO mass fraction is approximately 0.00421.

- e. Close the Surface Integrals dialog box.
- 9. Disable the prompt NOx mechanism in preparation for solving for thermal NOx only.

# **E** Setup $\rightarrow$ Models $\rightarrow$ Species $\rightarrow$ NOx $\stackrel{\text{D}}{\rightarrow}$ Edit...

- a. In the Formation tab, disable Prompt NOx.
- b. Click Apply and close the NOx Model dialog box.
- 10. Request 25 iterations.

### **□** Solution → Run Calculation

The solution will converge in approximately 6 iterations.

11. Review the thermal NOx solution by displaying the **contour-no-mass-fraction** contour definition for NO mass fraction (under the **Results/Graphics/Contours** tree branch) you created earlier (Figure 13.11: Contours of NO Mass Fraction—Thermal NOx Formation (p. 577)).

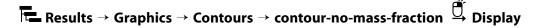
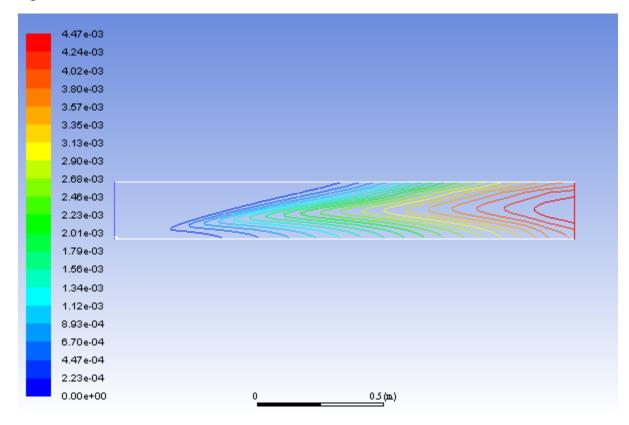


Figure 13.11: Contours of NO Mass Fraction—Thermal NOx Formation



Note that the concentration of NO is slightly lower without the prompt NOx mechanism.

12. Compute the average exit NO mass fraction with only thermal NOx formation.



### Tip

Follow the same procedure you used earlier for the calculation with both thermal and prompt NOx formation.

The **Mass-Weighted Average** field will show that the exit NO mass fraction with only thermal NOx formation (without prompt NOx formation) is approximately 0.004174.

13. Solve for prompt NOx production only.



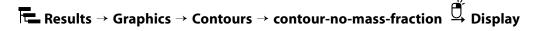
a. Disable Thermal NOx in the Pathways group box.

- b. Enable **Prompt NOx**.
- c. Click **Apply** and close the **NOx Model** dialog box.
- 14. Request 25 iterations.

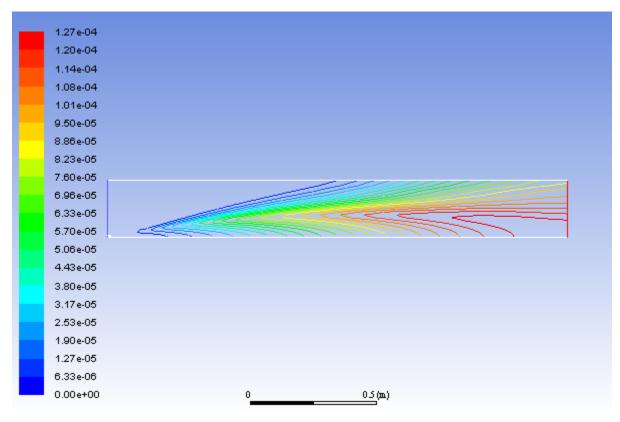
### **□** Solution → Run Calculation

The solution will converge in approximately 13 iterations.

15. Review the prompt NOx solution by displaying the **contour-no-mass-fraction** contour definition for NO mass fraction (under the **Results/Graphics/Contours** tree branch) (Figure 13.12: Contours of NO Mass Fraction—Prompt NOx Formation (p. 578)).







The prompt NOx mechanism is most significant in fuel-rich flames. In this case the flame is lean and prompt NO production is low.

16. Compute the average exit NO mass fraction only with prompt NOx formation.

### Postprocessing $\rightarrow$ Reports $\rightarrow$ Surface Integrals...

#### Tip

Follow the same procedure you used earlier for the calculation with both thermal and prompt NOx formation.

The **Mass-Weighted Average** field will show that the exit NO mass fraction with only prompt NOx formation is approximately 9.975e-05.

#### Note

The individual thermal and prompt NO mass fractions do not add up to the levels predicted with the two models combined. This is because reversible reactions are involved. NO produced in one reaction can be destroyed in another reaction.

17. Use a custom field function to compute NO parts per million (ppm).

The NOppm will be computed from the following equation:

$$NOppm = \frac{NOmole fraction \times 10^6}{1 - H_2Omole fraction}$$

(13.4)

#### Note

This is the dry ppm. Therefore, the value is normalized by removing the water mole fraction in the denominator.

User Defined $\rightarrow$ Field Functions $\rightarrow$ Cust	tom
---	-----

De	finition	d Function Ca	ilculator 0 ^ 6 / (1 - m	olef-h2o)				×
	+ INV 0 5 (	- sin 1 6 )	X Cos 2 7 PI	/ tan 3 8 e	y^x n 4 9	ABS log10 SQRT CE/C DEL	Select Operand Field Functions from Field Functions Species Mole fraction of h2o Select	•
Ne	w Function	Name no-ppn	n	Defin	e) (Manage.	Close H	lelp	

- a. Select NOx... and Mole fraction of Pollutant no from the Field Functions drop-down lists, and click the Select button to enter molef-pollut-pollutant-0 in the Definition field.
- b. Click the appropriate calculator buttons to enter

#### \*10^6/(1-

in the **Definition** field, as shown in the previous dialog box.

### Tip

If you make a mistake, click the **DEL** button on the calculator pad to delete the last item you added to the function definition.

For more explicit instructions on using the **Custom Field Function** calculator buttons, see Introduction to Using ANSYS Fluent: Fluid Flow and Heat Transfer in a Mixing Elbow (p. 121).

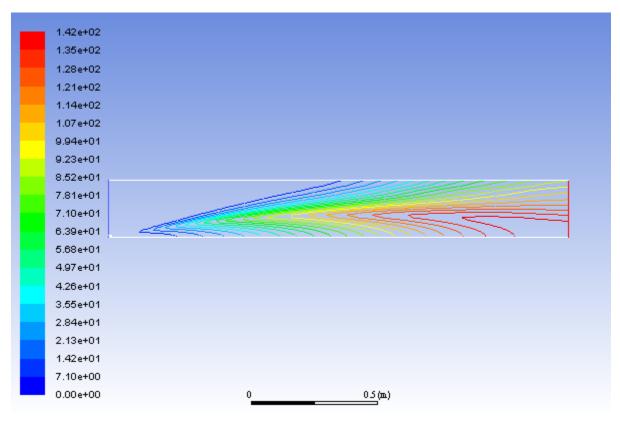
- c. Select **Species...** and **Mole fraction of h2o** from the **Field Functions** drop-down lists, and click the **Select** button to enter **molef-h2o** in the **Definition** field.
- d. Click the ) button to complete the field function.
- e. Enter no-ppm for New Function Name.
- f. Click **Define** to add the new field function to the variable list and close the **Custom Field Function Calculator** dialog box.
- 18. Display contours of NO ppm (Figure 13.13: Contours of NO ppm Prompt NOx Formation (p. 581)).

Postprocessing  $\rightarrow$  Graphics  $\rightarrow$  Contours  $\rightarrow$  Edit...

a. Select Custom Field Functions... and no-ppm from the Contours of drop-down lists.

Scroll up the list to find Custom Field Functions....

b. Click **Display** and close the **Contours** dialog box.



### Figure 13.13: Contours of NO ppm — Prompt NOx Formation

The contours closely resemble the mass fraction contours (Figure 13.12: Contours of NO Mass Fraction—Prompt NOx Formation (p. 578)), as expected.

## 13.6. Summary

In this tutorial you used ANSYS Fluent to model the transport, mixing, and reaction of chemical species. The reaction system was defined by using a mixture-material entry in the ANSYS Fluent database. The procedures used here for simulation of hydrocarbon combustion can be applied to other reacting flow systems.

The NOx production in this case was dominated by the thermal NO mechanism. This mechanism is very sensitive to temperature. Every effort should be made to ensure that the temperature solution is not overpredicted, since this will lead to unrealistically high predicted levels of NO.

## **13.7. Further Improvements**

Further improvements can be expected by including the effects of intermediate species and radiation, both of which will result in lower predicted combustion temperatures.

The single-step reaction process used in this tutorial cannot account for the moderating effects of intermediate reaction products, such as CO and  $H_2$ . Multiple-step reactions can be used to address these species. If a multi-step Magnussen model is used, considerably more computational effort is required to solve for the additional species. Where applicable, the nonpremixed combustion model can be used to account for intermediate species at a reduced computational cost.

For more details on the nonpremixed combustion model, see the Fluent User's Guide.

Radiation heat transfer tends to make the temperature distribution more uniform, thereby lowering the peak temperature. In addition, radiation heat transfer to the wall can be very significant (especially here, with the wall temperature set at 300 K). The large influence of radiation can be anticipated by computing the Boltzmann number for the flow:

$$Bo = \frac{\left(\rho U C_p\right)_{inlet}}{\sigma T_{AF}^3} \sim \frac{\text{convection}}{\text{radiation}}$$

where  $\sigma$  is the Boltzmann constant (5.729 ×10<sup>-8</sup>  $W/m^2-K^4$ ) and  $T_{AF}$  is the adiabatic flame temperature. For a quick estimate, assume  $\rho=1 \ kg/m^3$ ,  $U=0.5 \ m/s$ , and  $C_p=1000 \ J/kg-K$  (the majority of the inflow is air). Assume  $T_{AF}=2000 \ K$ . The resulting Boltzmann number is Bo = 1.09, which shows that radiation is of approximately equal importance to convection for this problem.

For details on radiation modeling, see the Fluent User's Guide.

This tutorial guides you through the steps to reach an initial set of solutions. You may be able to obtain a more accurate solution by using an appropriate higher-order discretization scheme and by adapting the mesh. Mesh adaption can also ensure that the solution is independent of the mesh. These steps are demonstrated in Introduction to Using ANSYS Fluent: Fluid Flow and Heat Transfer in a Mixing Elbow (p. 121).

# **Chapter 14: Using the Non-Premixed Combustion Model**

This tutorial is divided into the following sections:

14.1. Introduction
14.2. Prerequisites
14.3. Problem Description
14.4. Setup and Solution
14.5. Summary
14.6. References
14.7. Further Improvements

# 14.1. Introduction

The goal of this tutorial is to accurately model the combustion processes in a 300 KW BERL combustor. The reaction can be modeled using either the species transport model or the non-premixed combustion model. In this tutorial you will set up and solve a natural gas combustion problem using the non-premixed combustion model for the reaction chemistry.

This tutorial demonstrates how to do the following:

- Define inputs for modeling non-premixed combustion chemistry.
- Prepare the PDF table in ANSYS Fluent.
- Solve a natural gas combustion simulation problem.
- Use the Discrete Ordinates (DO) radiation model for combustion applications.
- Use the k- $\varepsilon$  turbulence model.

The non-premixed combustion model uses a modeling approach that solves transport equations for one or two conserved scalars (mixture fractions). Multiple chemical species, including radicals and intermediate species, may be included in the problem definition. Their concentrations will be derived from the predicted mixture fraction distribution.

Property data for the species are accessed through a chemical database, and turbulence-chemistry interaction is modeled using a  $\beta$ -function for the PDF. For details on the non-premixed combustion modeling approach, the Fluent User's Guide.

# 14.2. Prerequisites

This tutorial is written with the assumption that you have completed one or more of the introductory tutorials found in this manual:

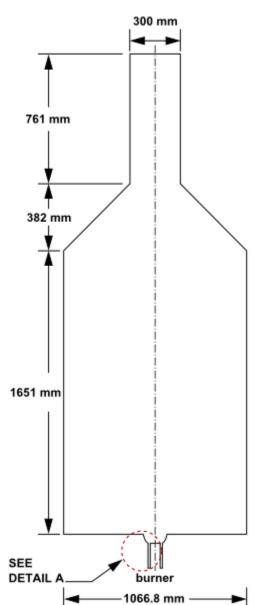
- Introduction to Using ANSYS Fluent in ANSYS Workbench: Fluid Flow and Heat Transfer in a Mixing Elbow (p. 1)
- Parametric Analysis in ANSYS Workbench Using ANSYS Fluent (p. 73)

• Introduction to Using ANSYS Fluent: Fluid Flow and Heat Transfer in a Mixing Elbow (p. 121)

and that you are familiar with the ANSYS Fluent tree and ribbon structure. Some steps in the setup and solution procedure will not be shown explicitly.

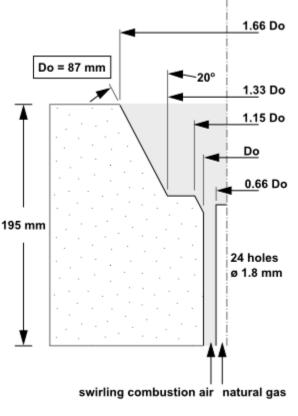
# 14.3. Problem Description

The flow considered is an unstaged natural gas flame in a 300 kW swirl-stabilized burner. The furnace is vertically-fired and of octagonal cross-section with a conical furnace hood and a cylindrical exhaust duct. The furnace walls are capable of being refractory-lined or water-cooled. The burner features 24 radial fuel ports and a bluff centerbody. Air is introduced through an annular inlet and movable swirl blocks are used to impart swirl. The combustor dimensions are described in Figure 14.1: Problem Description (p. 584), and Figure 14.2: Close-Up of the Burner (p. 585) shows a close-up of the burner assuming 2D axisymmetry. The boundary condition profiles, velocity inlet boundary conditions of the gas, and temperature boundary conditions are based on experimental data [1].



### Figure 14.1: Problem Description





DETAIL A

# 14.4. Setup and Solution

The following sections describe the setup and solution steps for this tutorial:

- 14.4.1. Preparation
- 14.4.2. Reading and Checking the Mesh
- 14.4.3. Specifying Solver and Analysis Type
- 14.4.4. Specifying the Models
- 14.4.5. Defining Materials and Properties
- 14.4.6. Specifying Boundary Conditions
- 14.4.7. Specifying Operating Conditions
- 14.4.8. Obtaining Solution
- 14.4.9. Postprocessing
- 14.4.10. Energy Balances Reporting

## 14.4.1. Preparation

To prepare for running this tutorial:

1. Set up a working folder on the computer you will be using.

2. Go to the ANSYS Customer Portal, https://support.ansys.com/training.

#### Note

If you do not have a login, you can request one by clicking **Customer Registration** on the log in page.

- 3. Enter the name of this tutorial into the search bar.
- 4. Narrow the results by using the filter on the left side of the page.
  - a. Click ANSYS Fluent under Product.
  - b. Click **18.0** under **Version**.
- 5. Select this tutorial from the list.
- 6. Click the **non\_premix\_combustion\_R180.zip** link to download the input files.
- 7. Unzip non\_premix\_combustion\_R180.zip to your working folder.

The files, berl.msh and berl.prof, can be found in the non\_premix\_combustion folder, which will be created after unzipping the file.

The mesh file, berl.msh, is a quadrilateral mesh describing the system geometry shown in Figure 14.1: Problem Description (p. 584) and Figure 14.2: Close-Up of the Burner (p. 585).

8. Use Fluent Launcher to start the **2D** version of ANSYS Fluent.

Fluent Launcher displays your **Display Options** preferences from the previous session.

- 9. Ensure that the **Display Mesh After Reading** and **Workbench Color Scheme** options are enabled.
- 10. Enable **Double-Precision**.

For more information about Fluent Launcher, see starting ANSYS Fluent using the Fluent Launcher in the Fluent Getting Started Guide.

11. Ensure that the **Serial** processing option is selected.

## 14.4.2. Reading and Checking the Mesh

1. Read the mesh file berl.msh.

### File → Read → Mesh...

The ANSYS Fluent console will report that the mesh contains 9784 quadrilateral cells. A warning will be generated informing you to consider making changes to the zone type, or to change the problem definition to axisymmetric. You will change the problem to axisymmetric swirl in Specifying Solver and Analysis Type (p. 590).

2. Check the mesh.

## **Setting Up Domain** $\rightarrow$ Mesh $\rightarrow$ Check

ANSYS Fluent will perform various checks on the mesh and will report the progress in the console. Ensure that the reported minimum volume is a positive number.

3. Scale the mesh.

**Setting Up Domain**  $\rightarrow$  Mesh  $\rightarrow$  Scale...

Scale Mes	h			
-Domain Ext	tents			Scaling
Xmin (mm)	0	Xmax (mm)	2989	Onvert Units
Ymin (mm)	0	Ymax (mm)	533.4001	Specify Scaling Factors
				Mesh Was Created In
				mm
View Length	Unit In			Scaling Factors
mm	•			X 0.001
				Y 0.001
				Scale Unscale
		Clos	se Help	

a. Select **mm** from the **View Length Unit In** drop-down list.

All dimensions will now be shown in millimeters.

- b. Select **mm** from the **Mesh Was Created In** drop-down list in the **Scaling** group box.
- c. Click Scale and verify that the domain extents are as shown in the Scale Mesh dialog box.
- d. Close the Scale Mesh dialog box.
- 4. Check the mesh.

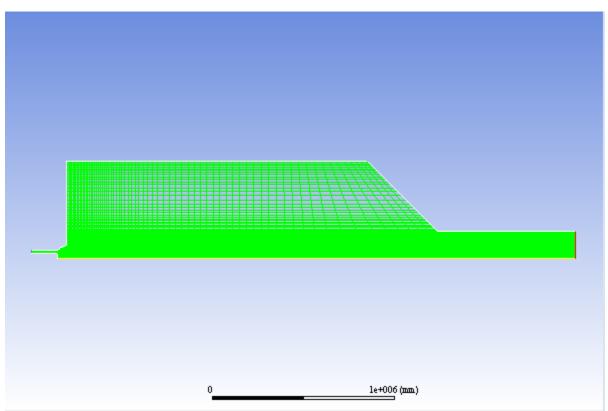
Setting Up Domain  $\rightarrow$  Mesh  $\rightarrow$  Check

#### Note

It is a good idea to check the mesh after you manipulate it (that is, scale, convert to polyhedra, merge, separate, fuse, add zones, or smooth and swap.) This will ensure that the quality of the mesh has not been compromised.

5. Examine the mesh (Figure 14.3: 2D BERL Combustor Mesh Display (p. 588)).

Figure 14.3: 2D BERL Combustor Mesh Display



Due to the mesh resolution and the size of the domain, you may find it more useful to display just the outline, or to zoom in on various portions of the mesh display.

### Extra

You can use the mouse zoom button (middle button, by default) to zoom in to the display and the mouse probe button (right button, by default) to find out the boundary zone labels. The zone labels will be displayed in the console.

6. Mirror the display about the symmetry plane.

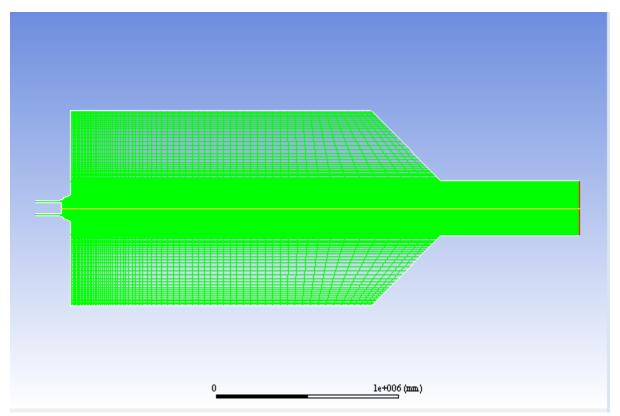
**Results**  $\rightarrow$  Graphics  $\stackrel{\frown}{\hookrightarrow}$  Views...

<b>E</b> Views		
Views back front Save Name view-0	Actions Default Auto Scale Previous Save Delete Read Write	Mirror Planes [1/1]           axis-2         Define Plane         Periodic Repeats         Define
Apph	Camera	Close Help

- a. Select axis-2 from the Mirror Planes selection list.
- b. Click **Apply** and close the **Views** dialog box.

The full geometry is displayed, as shown in Figure 14.4: 2D BERL Combustor Mesh Display Including the Symmetry Plane (p. 589)





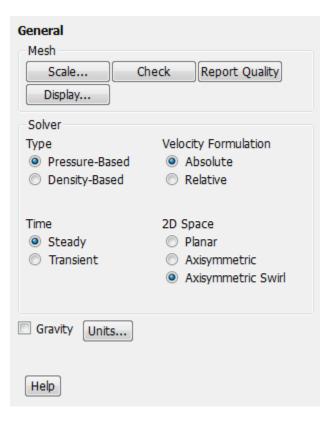
# 14.4.3. Specifying Solver and Analysis Type

1. Retain the default settings of pressure-based steady-state solver in the **Solver** group box.

## E Setup $\rightarrow$ General

The non-premixed combustion model is available only with the pressure-based solver.

2. Change the spatial definition to axisymmetric swirl by selecting **Axisymmetric Swirl** in the **2D Space** list.



# 14.4.4. Specifying the Models

1. Enable the **Energy Equation**.

Setting Up Physics → Models → Energy

Since heat transfer occurs in the system considered here, you will have to solve the energy equation.

2. Select the standard k-epsilon turbulence model.

Setting Up Physics  $\rightarrow$  Models  $\rightarrow$  Viscous...

Viscous Model	<b>—</b> ×
Model	Model Constants
Inviscid	Cmu
🔘 Laminar	0.09
Spalart-Allmaras (1 eqn)	C1-Epsilon
k-epsilon (2 eqn)	1.44
<ul> <li>k-omega (2 eqn)</li> <li>Transition k-kl-omega (3 eqn)</li> </ul>	C2-Epsilon
<ul> <li>Transition SST (4 eqn)</li> </ul>	1.92
Reynolds Stress (7 eqn)	TKE Prandtl Number
$\odot$ Scale-Adaptive Simulation (SAS)	1
Detached Eddy Simulation (DES)	TDR Prandtl Number
k-epsilon Model	1.3
Standard	
© RNG	User-Defined Functions
🔘 Realizable	Turbulent Viscosity
Near-Wall Treatment	none 🔻
Standard Wall Functions	Prandtl Numbers
Scalable Wall Functions	TKE Prandtl Number
<ul> <li>Non-Equilibrium Wall Functions</li> <li>Enhanced Wall Treatment</li> </ul>	none
Menter-Lechner	TDR Prandtl Number
User-Defined Wall Functions	none 💌 🗉
Options	Energy Prandtl Number
Viscous Heating	none
Production Kato-Launder	Wall Prandtl Number
Production Limiter	none 👻 🗸
OK (	Cancel Help

a. Select k-epsilon (2eqn) in the Model list.

For axisymmetric swirling flow, the RNG k-epsilon model can also be used, but for this case you will retain the default **Standard**, *k*-epsilon model.

- b. Retain the default Standard Wall Functions in the Near-Wall Treatment group box.
- c. Click **OK** to accept all other default settings and close the **Viscous Model** dialog box.
- 3. Select the **Discrete Ordinates (DO)** radiation model.

The DO radiation model provides a high degree of accuracy, but it can be CPU intensive. In cases where the computational expense of the DO model is deemed too great, the P1 model may provide an acceptable solution more quickly.

For details on the different radiation models available in ANSYS Fluent, see the Fluent User's Guide.

Setting Up Physics  $\rightarrow$  Models  $\rightarrow$  Radiation...

#### a. Select Discrete Ordinates (DO) in the Model list.

The dialog box will expand to show related inputs.

Radiation Model		×
Model Off Rosseland P1 Discrete Transfer (DTRM) Surface to Surface (S2S) Discrete Ordinates (DO)	Iteration Parameters Energy Iterations per Radiation Iteration 1 Angular Discretization Theta Divisions 2 Phi Divisions 2 Theta Pixels 1	
DO/Energy Coupling	Phi Pixels 1	_
	OK Cancel Help	

- b. Enter 1 for Energy Iterations per Radiation Iteration.
- c. Click **OK** to accept all other default settings and close the **Radiation Model** dialog box.

The ANSYS Fluent console lists the properties that are required for the model you have enabled. An *Information* dialog box opens, reminding you to confirm the property values.

💶 Inform	nation
i	Available material properties or methods have changed. Please confirm the property values before continuing.
	ОК

- d. Click **OK** to close the **Information** dialog box.
- 4. Select the Non-Premixed Combustion model.



Species Model Model Off Species Transport Non-Premixed Combustion Premixed Combustion Partially Premixed Combustion Composition PDF Transport	Chemistry Boundary Control Flamelet Table Properties Premix			
	State Relation       Energy Treatment       Stream Options         Image: Chemical Equilibrium       Adiabatic       Secondary Stream         Image: Steady Diffusion Flamelet       Non-Adiabatic       Empirical Fuel Stream         Image: Disel Unsteady Flamelet       Flamelet Generated Manifold       Flamelet Generated Manifold			
PDF Options ✓ Inlet Diffusion Compressibility Effects	Model Settings Operating Pressure (pascal) 101325 Fuel Stream Rich Flamability Limit 0.064 Coal Calculator			
	Thermodynamic Database File Name			
	\\cpropep\data\\thermo.db			
	OK Apply Cancel Help			

a. Select Non-Premixed Combustion in the Model list.

The dialog box will expand to show the related inputs. You will use this dialog box to create the PDF table.

When you use the non-premixed combustion model, you need to create a PDF table. This table contains information on the thermo-chemistry and its interaction with turbulence. ANSYS Fluent interpolates the PDF during the solution of the non-premixed combustion model.

b. Enable Inlet Diffusion in the PDF Options group box.

The **Inlet Diffusion** option enables the mixture fraction to diffuse out of the domain through inlets and outlets.

- c. Define chemistry models.
  - i. Retain the default selection of the **Chemical Equilibrium** state relation and the **Non-Adiabatic** energy treatment.

In most non-premixed combustion simulations, the **Chemical Equilibrium** model is recommended. The **Steady Diffusion Flamelet** option can model local chemical non-equilibrium due to turbulent strain.

- ii. Retain the default value for Operating Pressure.
- iii. Enter 0.064 for Fuel Stream Rich Flammability Limit.

The **Fuel Stream Rich Flammability Limit** enables you to perform a "partial equilibrium" calculation, suspending equilibrium calculations when the mixture fraction exceeds the specified rich limit. This increases the efficiency of the PDF calculation, allowing you to bypass the complex equilibrium calculations in the fuel-rich region. This is also more physically realistic than the assumption of full equilibrium. For combustion cases, a value 10% – 50% larger than the stoichiometric mixture fraction can be used for the rich flammability limit of the fuel stream. In this case, the stoichiometric fraction is 0.058, therefore a value that is 10% greater is 0.064.

d. Click the Boundary tab to add and define the boundary species.

Species Model				×
Model O Off	Chemistry Boundary Control	Flamelet Table	Properties	Premix
<ul> <li>Species Transport</li> <li>Non-Premixed Combustion</li> <li>Premixed Combustion</li> <li>Partially Premixed Combustion</li> <li>Composition PDF Transport</li> </ul>	Species	Fuel	Oxid	
	ch4		0.965	0
	h2		0	0
	jet-a <g></g>		0	0
	n2		0.013	0.78992
	02		0	0.21008
	c2h6		0.017	0
	c3h8		0.001	0
	c4h10		0.001	0
	co2		0.003	0
PDF Options           Inlet Diffusion           Compressibility Effects	Boundary Species co2 Add Remove List Available Species	Temperature Fuel (k) 315 Oxid (k) 315	Specify Sp Mass Fr Mole Fr	action
	OK Apply Cancel He	lp		

i. Enter c2h6 in the Boundary Species text-entry field and click Add.

The **c2h6** species appears at the bottom of the table.

ii. Similarly, add c3h8, c4h10, and co2.

All the added species will appear in the table.

- iii. Select Mole Fraction in the Specify Species in list.
- iv. Retain the default values for **n2** and **o2** for **Oxid**.

The oxidizer (air) consists of 21%  $O_{\scriptscriptstyle 2}$  and 79%  $2N_{\scriptscriptstyle 2}$  by volume.

v. Specify the fuel composition by entering the following values for **Fuel**:

The fuel composition is entered in mole fractions of the species, c2h6, c3h8, c4h10, and co2.

Species	Mole Fraction
ch4	0.965
n2	0.013

Species	Mole Fraction		
c2h6	0.017		
c3h8	0.001		
c4h10	0.001		
co2	0.003		

### Tip

Scroll down to see all the species.

#### Note

All boundary species with a mass or mole fraction of zero will be ignored.

- vi. Enter 315 K for Fuel and Oxid in the Temperature group box.
- e. Click the **Control** tab and retain default species to be excluded from the equilibrium calculation.
- f. Click the **Table** tab to specify the table parameters and calculate the PDF table.

Species Model	×
Model Off	Chemistry Boundary Control Flamelet Table Properties Premix
<ul> <li>On</li> <li>Species Transport</li> <li>Non-Premixed Combustion</li> <li>Premixed Combustion</li> <li>Partially Premixed Combustion</li> <li>Composition PDF Transport</li> </ul>	Table Parameters Initial Number of Grid Points 15 🔶 Maximum Number of Grid Points 200 🔶 Maximum Change in Value Ratio 0.25 Maximum Change in Slope Ratio 0.25
PDF Options       Inlet Diffusion	Maximum Number of Species 20 Minimum Temperature (k) 298 Automated Grid Refinement Calculate PDF Table Display PDF Table
Compressibility Effects	Calculate PDF Table

- i. Ensure that Automated Grid Refinement is enabled.
- ii. Retain the default values for all the parameters in the Table Parameters group box.

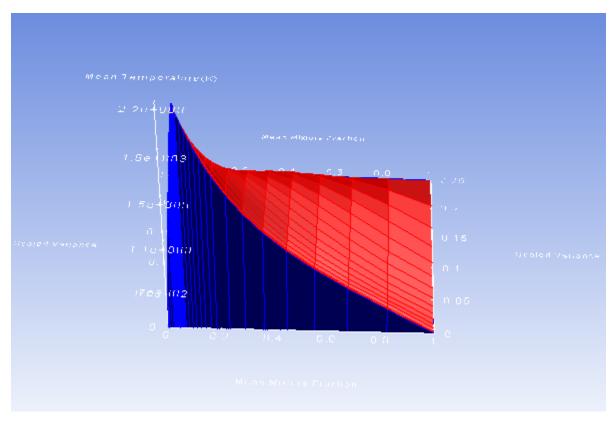
The maximum number of species determines the number of most preponderant species to consider after the equilibrium calculation is performed.

- iii. Click Calculate PDF Table to compute the non-adiabatic PDF table.
- iv. Click the **Display PDF Table...** button.

PDF Data Type Nonadiabatic Table (Two Stre	ams)				
Plot Variable					
Mean Temperature (K)					
Plot Type ● 3D Surface ○ 2D Curve on 3D Surface		Numbers Box To File			
Surface Parameters Constant Value of	Slice by	Index			
	Index	Index Min	Max	Adiabatic	
<ul> <li>Mean Mixture Fraction</li> <li>Scaled Variance</li> </ul>	Value	13 🜩 1	25	13	
2					

- A. In the **PDF Table** dialog box, retain the default parameters and click **Display** (Figure 14.5: Non-Adiabatic Temperature Look-Up Table on the Adiabatic Enthalpy Slice (p. 596)).
- B. Close the **PDF Table** dialog box.

Figure 14.5: Non-Adiabatic Temperature Look-Up Table on the Adiabatic Enthalpy Slice



The 3D look-up tables are reviewed on a slice-by-slice basis. By default, the slice selected corresponds to the adiabatic enthalpy values. You can also select other slices of constant enthalpy for display.

The maximum and minimum values for mean temperature and the corresponding mean mixture fraction is also reported in the console. The maximum mean temperature is reported as 2246 K at a mean mixture fraction of 0.058.

g. Save the PDF output file (berl.pdf).

### **File** $\rightarrow$ Write $\rightarrow$ PDF...

- i. Retain berl.pdf for PDF File name.
- ii. Click **OK** to write the file.

By default, the file will be saved as formatted (ASCII, or text). To save a binary (unformatted) file, enable the **Write Binary Files** option in the **Select File** dialog box.

h. Click OK to close the Species Model dialog box.

## 14.4.5. Defining Materials and Properties

1. Specify the continuous phase (pdf-mixture) material.

💶 Setup 🕯	<b>↓</b> → N	Naterials $\rightarrow$		pdf-mixture → Create	e/Edit
-----------	--------------	-------------------------	--	----------------------	--------

Create/Edit Materials		<b></b>		
Name	Material Type	Order Materials by		
pdf-mixture	mixture			
Chemical Formula	Fluent Mixture Materials	Chemical Formula		
	pdf-mixture •	Fluent Database		
	Mixture	User-Defined Database		
	none	User-Defined Database		
Properties				
Thermal Conductivity (w/m-k) constant	▼ Edit ^			
0.0454	0.0454			
Viscosity (kg/m-s) constant				
1.72e-05				
Absorption Coefficient (1/m) user-defined-wsggm	▼ Edit E			
Scattering Coefficient (1/m) constant	▼ Edit			
0	-			
Change/Create Delete Close Help				

All thermodynamic data for the continuous phase, including density, specific heat, and formation enthalpies are extracted from the chemical database when the non-premixed combustion model is used. These properties are transferred to the **pdf-mixture** material, for which only transport properties, such as viscosity and thermal conductivity, need to be defined. The **Density** and **Cp (Specific Heat)** laws cannot be altered: these properties are stored in the non-premixed combustion look-up tables.

Under **Properties**, you can click the **View...** button next to **Mixture Species** to view the species included in the **pdf-mixture** material. These are the species included during the system chemistry setup.

a. Under Properties, from the Absorption Coefficient drop-down list, select wsggm-domain-based.

Tip

Scroll down to view the Absorption Coefficient option.

This specifies a composition-dependent absorption coefficient, using the weighted-sum-of-gray-gases model. WSGGM-domain-based is a variable coefficient that uses a length scale, based on the geometry of the model.

For more details, see radiation in combusting flows in the Fluent Theory Guide.

b. Click Change/Create and close the Create/Edit Materials dialog box.

ANSYS Fluent uses the gas law to compute the mixture density and a mass-weighted mixing law to compute the mixture  $C_p$ . When the non-premixed combustion model is used, do not alter the properties of the individual species. This will create an inconsistency with the PDF look-up table.

## 14.4.6. Specifying Boundary Conditions

1. Read the boundary conditions profile file.

### File $\rightarrow$ Read $\rightarrow$ Profile...

- a. Select berl.prof from the Select File dialog box.
- b. Click OK.

The CFD solution for reacting flows can be sensitive to the boundary conditions, in particular the incoming velocity field and the heat transfer through the walls. Here, you will use profiles to specify the velocity at **air-inlet-4**, and the wall temperature for **wall-9**. The latter approach of fixing the wall temperature to measurements is common in furnace simulations, to avoid modeling the wall convective and radiative heat transfer. The data used for the boundary conditions was obtained from experimental data [1].

2. Set the boundary conditions for the pressure outlet (poutlet-3).

**Setup**  $\rightarrow$  **Conditions**  $\rightarrow$  **Edit...** 

Pressure Outle	t					×	
Zone Name							
poutlet-3							
Momentum	Thermal Radiatio	n Species	DPM	Multiphase	Potential	UDS	
Bac	Backflow Reference Frame Absolute						
	Gauge Pressure (	pascal) 0		CO	nstant		
Backflow Direction	on Specification Meth	od Normal to E	Boundary				
Backflov	v Pressure Specificat	ion Total Press	ure			•	
🔲 Radial Equilibr	ium Pressure Distribu	tion					
Average Pres	sure Specification						
🔲 Target Mass F	low Rate						
	Turbulence						
	Specification Metho	d Intensity an	d Hydraul	ic Diameter		<b>_</b>	
	Backflow Turbulent Intensity (%) 5						
Backflow Hydraulic Diameter (mm) 600							
OK Cancel Help							

- a. In the **Turbulence** group, from the **Specification Method** drop-down list, select **Intensity and Hydraulic Diameter**.
- b. Retain 5% for **Backflow Turbulent Intensity**.
- c. Enter 600 mm for **Backflow Hydraulic Diameter**.
- d. Click the **Thermal** tab and enter 1300 K for **Backflow Total Temperature**.
- e. Click OK to close the Pressure Outlet dialog box.

The exit gauge pressure of zero defines the system pressure at the exit to be the operating pressure. The backflow conditions for scalars (temperature, mixture fraction, turbulence parameters) will be used only if flow is entrained into the domain through the exit. It is a good idea to use reasonable values in case flow reversal occurs at the exit at some point during the solution process.

3. Set the boundary conditions for the velocity inlet (air-inlet-4).

**E** Setup  $\rightarrow$  **\bigcirc** Boundary Conditions  $\rightarrow$  **\equiv** $air-inlet-4 \rightarrow$  Edit...

	Velocity Inlet			<b>×</b>			
	Zone Name						
	air-inlet-4						
	Momentum Thermal Radiation Species DPM Multi	phase	Potential	UDS			
	Velocity Specification Method Components			•			
	Reference Frame Absolute			•			
	Supersonic/Initial Gauge Pressure (pascal) 0 constant						
	Axial-Velocity (m/s)	vel-p	rof u	•			
	Radial-Velocity (m/s) 0	const	tant	•			
	Swirl-Velocity (m/s)	vel-p	rof w	•			
	Swirl Angular Velocity (rad/s)	0		P			
	Turbulence						
	Specification Method Intensity and Hydraulic Diamet	ter		<b>_</b>			
	Turbulent Intensity (%) 17						
	Hydraulic Diameter (mm) 29						
	OK Cancel Help						

- a. From the Velocity Specification Method drop-down list, select Components.
- b. From the Axial-Velocity drop-down list, select vel-prof u.
- c. From the Swirl-Velocity drop-down list, select vel-prof w.
- d. In the **Turbulence** group, from the **Specification Method** drop-down list, select **Intensity and Hydraulic Diameter**.
- e. Enter 17% for **Turbulent Intensity**.
- f. Enter 29 mm for Hydraulic Diameter.

Turbulence parameters are defined based on intensity and length scale. The relatively large turbulence intensity of 17% may be typical for combustion air flows.

- g. Click the Thermal tab and enter 312 K for Temperature.
- h. Click the **Species** tab. For the non-premixed combustion calculation, you have to define the inlet **Mean Mixture Fraction** and **Mixture Fraction Variance**. In this case, the gas phase air inlet has a zero mixture fraction. Therefore, you can retain the zero default settings.
- i. Click **OK** to close the **Velocity Inlet** dialog box.
- 4. Set the boundary conditions for the velocity inlet (**fuel-inlet-5**).

**F**Setup  $\rightarrow$  **\bigcirc** Boundary Conditions  $\rightarrow \stackrel{\frown}{=}$  fuel-inlet-5  $\rightarrow$  Edit...

	Velocity Inlet				<b>—</b>	
į	Zone Name					
	fuel-inlet-5					
	Momentum Thermal Radiation	Species DPM N	Multipha	se Potenti	al UDS	
	Velocity Specification Method Co	mponents			•	
	Reference Frame Ab	solute			•	
	Supersonic/Initial Gauge Pressure (pascal) 0 constant					
	Axial-Velocity (m/s)	0		onstant	•	
	Radial-Velocity (m/s)	157.25		onstant	<b>_</b>	
	Swirl-Velocity (m/s)	0	•	onstant	<b>_</b>	
	Sw	irl Angular Velocity (rad/	/s) 0		P	
	Turbulence					
	Specification Method Int	ensity and Hydraulic Dia	meter		•	
	Turbulent Intensity (%) 5					
	Hydraulic Diameter (mm) 1.8					
ļ						
	OK Cancel Help					

- a. From the Velocity Specification Method drop-down list, select Components.
- b. Enter 157.25 m/s for Radial-Velocity.
- c. In the **Turbulence** group, from the **Specification Method** drop-down list, select **Intensity and Hydraulic Diameter**.
- d. Retain 5% for **Turbulent Intensity**.
- e. Enter 1.8 mm for **Hydraulic Diameter**.

The hydraulic diameter has been set to twice the height of the 2D inlet stream.

- f. Click the **Thermal** tab and enter 308 K for **Temperature**.
- g. Click the Species tab and enter 1 for Mean Mixture Fraction for the fuel inlet.
- h. Click **OK** to close the **Velocity Inlet** dialog box.
- 5. Set the boundary conditions for **wall-6**.

**E** Setup  $\rightarrow$  **\bigcirc** Boundary Conditions  $\rightarrow$  **\stackrel{\frown}{=}** wall-6  $\rightarrow$  Edit...

💶 Wall			<b>—</b>		
Zone Name					
wall-6					
Adjacent Cell Zone					
fluid-15					
Momentum Thermal F	Radiation Species DPM M	ultiphase UDS Wall	Film Potential		
Thermal Conditions					
Heat Flux	Temperature (k	) 1370	constant 💌		
Temperature	Internal Emissivit	0.5	constant 👻		
<ul> <li>Convection</li> <li>Radiation</li> </ul>		Wall Thickness (mm) 0	P		
Mixed	Heat Generation Rate (w/m3		constant 🔹		
via System Coupling					
via Mapped Interface					
Material Name					
aluminum 💌	Edit				
	OK Cancel Help				

- a. Click the **Thermal** tab.
  - i. In the Thermal Conditions list, select Temperature.
  - ii. Enter 1370 K for Temperature.
  - iii. Enter 0.5 for Internal Emissivity.
- b. Click **OK** to close the **Wall** dialog box.
- 6. Similarly, set the boundary conditions for **wall-7** through **wall-13** using the following values:

Zone Name	Temperature	Internal Emissivity
wall-7	312	0.5
wall-8	1305	0.5
wall-9	temp-prof t (from the drop-down list)	0.5
wall-10	1100	0.5
wall-11	1273	0.5
wall-12	1173	0.5
wall-13	1173	0.5

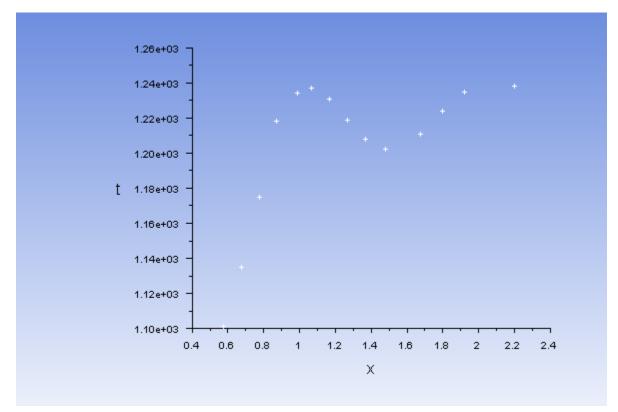
7. Plot the profile of temperature for the wall furnace (wall-9).

## Postprocessing $\rightarrow$ Plots $\rightarrow$ Profile Data...

Plot Profile Data		
Profile	Y Axis Function	X Axis Function
vel-prof	t	x
temp-prof		У
Pk	Axes Curves Close H	lelp
		-11

- a. From the **Profile** selection list, select **temp-prof**.
- b. Retain the selection of t and x from the Y Axis Function and X Axis Function selection lists, respectively.
- c. Click Plot (Figure 14.6: Profile Plot of Temperature for wall-9 (p. 603)).

Figure 14.6: Profile Plot of Temperature for wall-9



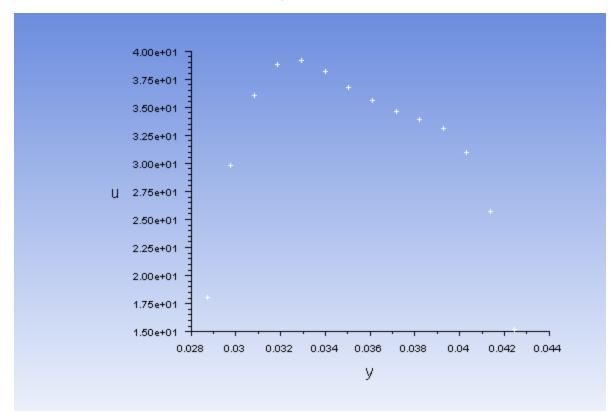
- 8. Plot the profiles of velocity for the swirling air inlet (air-inlet-4).
  - a. Plot the profile of axial-velocity for the swirling air inlet.

## Postprocessing $\rightarrow$ Plots $\rightarrow$ Profile Data...

Plot Profile Data					
Profile	Y Axis Function	X Axis Function			
vel-prof	u	x			
temp-prof	w	у			
Plot Axes Curves Close Help					

- i. From the **Profile** selection list, select **vel-prof**.
- ii. From the **Y Axis Function** selection list, retain the selection of **u**.
- iii. From the X Axis Function selection list, select y.
- iv. Click Plot (Figure 14.7: Profile Plot of Axial-Velocity for the Swirling Air Inlet (air-inlet-4) (p. 604)).

Figure 14.7: Profile Plot of Axial-Velocity for the Swirling Air Inlet (air-inlet-4)

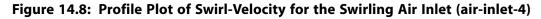


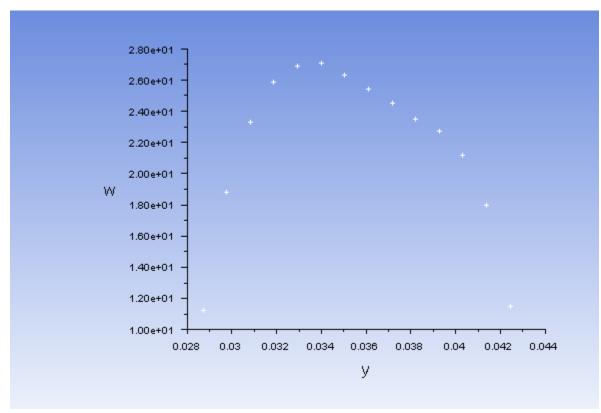
b. Plot the profile of swirl-velocity for swirling air inlet.

Postprocessing  $\rightarrow$  Plots  $\rightarrow$  Profile Data...

rofile	Y Axis Function	X Axis Function
/el-prof emp-prof	w	x y

- i. From the Profile selection list, retain the selection of vel-prof.
- ii. From the Y Axis Function selection list, select w.
- iii. From the X Axis Function selection list, retain the selection of y.
- iv. Click **Plot** (Figure 14.8: Profile Plot of Swirl-Velocity for the Swirling Air Inlet (air-inlet-4) (p. 606)) and close the **Plot Profile Data** dialog box.





# 14.4.7. Specifying Operating Conditions

- 1. Retain the default operating conditions.
  - **T** Setup  $\rightarrow$  **\bigcirc** Boundary Conditions  $\rightarrow$  Operating Conditions...

Operating Conditions	<b>-X</b> -			
Pressure	Gravity			
Operating Pressure (pascal)	Gravity			
101325 P				
Reference Pressure Location				
X (mm) 0				
Y (mm) 0				
Z (mm) 0				
OK Cancel Help				

The **Operating Pressure** was already set in the PDF table generation in Specifying the Models (p. 590).

## 14.4.8. Obtaining Solution

**Solving**  $\rightarrow$  Solution  $\rightarrow$  Methods...

1. Set the solution parameters.

Solution Methods	
Pressure-Velocity Coupling	
Scheme	
Coupled	•
Spatial Discretization	
Gradient	^
Least Squares Cell Based 🔹	
Pressure	=
PRESTO!	-
Momentum	
Second Order Upwind 🔹	
Swirl Velocity	
Second Order Upwind 🔹	
Turbulent Kinetic Energy	
First Order Upwind 🔹	_
Transient Formulation	
Non-Iterative Time Advancement	
Frozen Flux Formulation	
Pseudo Transient	
Warped-Face Gradient Correction	
High Order Term Relaxation Options	
Default	

- a. In the **Pressure-Velocity Coupling** group, from the **Scheme** drop-down list, select **Coupled**.
- b. In the Spatial Discretization group, from the Pressure drop-down list, select PRESTO!.
- c. Retain the other default selections and settings.
- 2. Set the solution controls.



	ourant Numbe	er	
70			
	Explicit Rela	exation Factors	
	Momentum	0.5	
	Pressure	0.5	
Under-	Relaxation Fac	tors	
Den	sity		
0.2			
Bod	y Forces		Ξ
0.8			
Swir	l Velocity		
0.9			
Turb	ulent Kinetic	Energy	
0.8			
Turb	ulent Dissipat	tion Rate	
0.8			
			Ŧ
Defau	lt		

- a. Enter 70 for Flow Courant Number.
- b. Set the following parameters in the **Under-Relaxation Factors** group box:

Under-Relaxation Factor	Value
Density	0.2
Body Forces	0.8

The default under-relaxation factors are considered to be too aggressive for reacting flow cases with high swirl velocity.

3. Enable the display of residuals during the solution process.

# Solving $\rightarrow$ Reports $\rightarrow$ Residuals...

ſ					
Residual Monitors					×
Options	Equations				
Print to Console	Residual	Monitor	r Check Converge	ence Absolute Criteria	<u>^</u>
V Plot	continuity	V	$\checkmark$	0.001	E
Window	x-velocity	<b>V</b>	$\checkmark$	0.001	
1 Curves Axes	y-velocity	<b>V</b>	$\checkmark$	0.001	]
Iterations to Plot	swirl	V		0.001	
1000 🗢	Residual Values		(ma)	Convergence Criterio	n 🖾 I
	Normalize		Iterations	absolute	•
Iterations to Store			5		
1000 🗢	Scale			Convergence Condit	ions
Compute Local Scale					
OK Plot Renormalize Cancel Help					

- a. Ensure that the **Plot** is enabled in the **Options** group box.
- b. Click **OK** to close the **Residual Monitors** dialog box.
- 4. Initialize the flow field.



Initialization						
Method		Patch				
O Hybrid	More Settings	Reset Statistics				
Standard	Options		t = 0			
		Reset DPM	Initialize			

- a. Retain the **Method** at the default of **Hybrid** in the **Initialization** group.
- b. Click Initialize.
- 5. Save the case file (berl-1.cas.gz).

```
File \rightarrow Write \rightarrow Case...
```

6. Start the calculation by requesting 1500 iterations.



Run Calculation				
Update Dynamic Mesh				
Input Summary	No. of Iterations 1500	•	-/	
Advanced	Check Case		Calculate	

The solution will converge in approximately 1060 iterations.

7. Save the converged solution (berl-1.dat.gz).

**File**  $\rightarrow$  Write  $\rightarrow$  Data...

## 14.4.9. Postprocessing

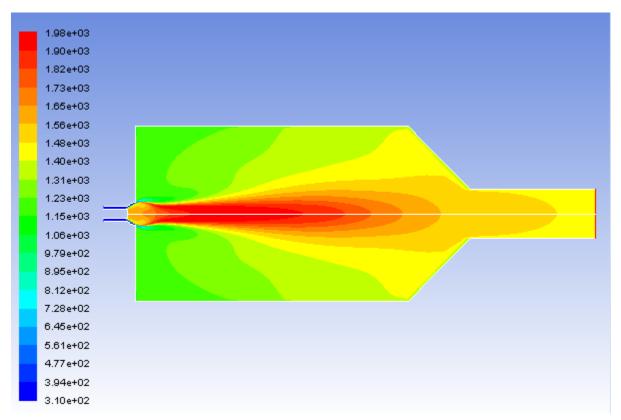
1. Display the predicted temperature field (Figure 14.9: Temperature Contours (p. 611)).

Postprocessing  $\rightarrow$  Graphics  $\rightarrow$  Contours  $\rightarrow$  Edit...

Contours	
Options Filled	Contours of Temperature
Node Values Global Range	Static Temperature 🔹
<ul> <li>Global Range</li> <li>Auto Range</li> </ul>	Min (k) Max (k)
Clip to Range	309.9886 1982.591
Draw Profiles Draw Mesh	Surfaces Filter Text
Coloring Banded Smooth Levels Setup 20 1	air-inlet-4 axis-2 default-interior fuel-inlet-5 poutlet-3 wall-10 wall-11 ▼
	Display Compute Close Help

- a. Enable **Filled** in the **Options** group box.
- b. Select Temperature... and Static Temperature from the Contours of drop-down lists.
- c. Click Display.

The peak temperature in the system is 1981 K.

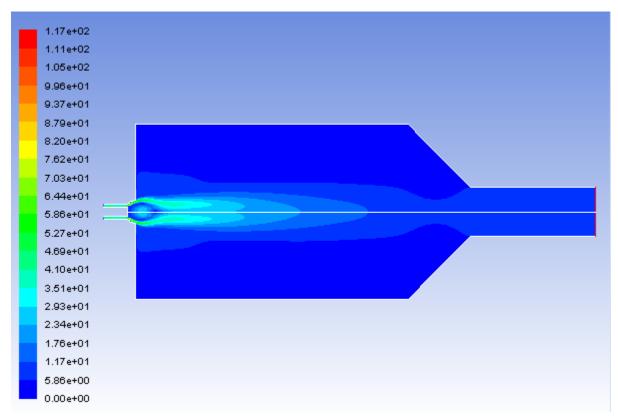


#### Figure 14.9: Temperature Contours

2. Display contours of velocity (Figure 14.10: Velocity Contours (p. 612)).

Postprocessing  $\rightarrow$  Graphics  $\rightarrow$  Contours  $\rightarrow$  Edit...

- a. Select Velocity... and Velocity Magnitude from the Contours of drop-down lists.
- b. Click **Display**.



### Figure 14.10: Velocity Contours

3. Display the contours of mass fraction of O<sub>2</sub> (Figure 14.11: Contours of Mass Fraction of o2 (p. 613)).

Postprocessing  $\rightarrow$  Graphics  $\rightarrow$  Contours  $\rightarrow$  Edit...

- a. Select Species... and Mass fraction of o2 from the Contours of drop-down lists.
- b. Click **Display** and close the **Contours** dialog box.

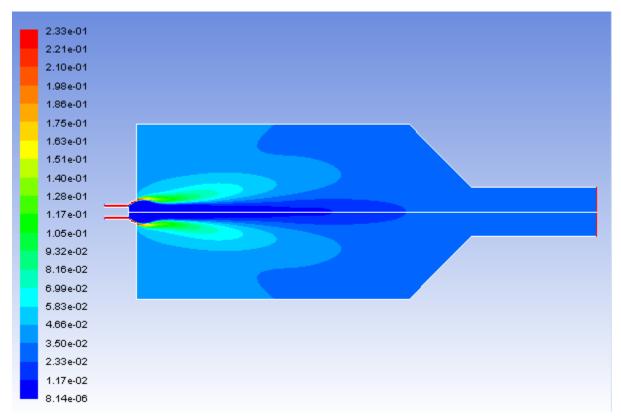
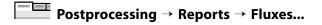


Figure 14.11: Contours of Mass Fraction of o2

## 14.4.10. Energy Balances Reporting

ANSYS Fluent can report the overall energy balance and details of the heat and mass transfer.

1. Compute the gas phase mass fluxes through the domain boundaries.



Flux Reports		<b>—</b>
Options     Mass Flow Rate	Boundaries Filter Text	Results
<ul> <li>Total Heat Transfer Rate</li> <li>Total Sensible Heat Transfer Rate</li> </ul>	air-inlet-4 A	0.1194955072357111
Radiation Heat Transfer Rate	default-interior fuel-inlet-5 poutlet-3 wall-10 wall-11 wall-12 wall-13 wall-6 wall-7 wall 9	0.006312730064526788 -0.1258082959244725
	۲. (۲. (۲. (۲. (۲. (۲. (۲. (۲. (۲. (۲. (	٠ ٢
Save Output Parameter		Net Results (kg/s)
		-5.862423e-08
G	Compute Write Close Help	

- a. Retain the default selection of Mass Flow Rate in the Options group box.
- b. Select air-inlet-4, fuel-inlet-5, and poutlet-3 from the Boundaries selection list.
- c. Click Compute.

The net mass imbalance should be a small fraction of the total flux through the system. If a significant imbalance occurs, you should decrease your residual tolerances by at least an order of magnitude and continue iterating.

2. Compute the fluxes of heat through the domain boundaries.

Postprocessing  $\rightarrow$  Reports  $\rightarrow$  Fluxes...

- a. Select Total Heat Transfer Rate in the Options group box.
- b. Select all the zones from the **Boundaries** selection list.
- c. Click **Compute**, examine the resulting values, and close the **Flux Reports** dialog box.

The value will be displayed in the console. Positive flux reports indicate heat addition to the domain. Negative values indicate heat leaving the domain. Again, the net heat imbalance should be a small fraction (for example, 0.5% or less) of the total energy flux through the system. The reported value may change for different runs.

3. Compute the mass weighted average of the temperature at the pressure outlet.

Postprocessing → Reports → Surface Integrals...

Surface Integrals	
Report Type	Field Variable
Mass-Weighted Average 🔹	Temperature
Custom Vectors	Static Temperature 👻
Vectors of	Surfaces Filter Text
Custom Vectors	air-inlet-4
	axis-2
Save Output Parameter	default-interior
	fuel-inlet-5
	poutlet-3
	wall-10
	wall-11
	wall-12
	wall-13
	wall-6
	Mass-Weighted Average (k)
	1464.727
Compute	/rite Close Help

- a. Select Mass-Weighted Average from the Report Type drop-down list.
- b. Select Temperature... and Static Temperature from the Field Variable drop-down lists.
- c. Select poutlet-3 from the Surfaces selection list.
- d. Click Compute.

A value of approximately 1464 K will be displayed in the console.

e. Close the Surface Integrals dialog box.

## 14.5. Summary

In this tutorial you learned how to use the non-premixed combustion model to represent the gas phase combustion chemistry. In this approach the fuel composition was defined and assumed to react according to the equilibrium system data. This equilibrium chemistry model can be applied to other turbulent, diffusion-reaction systems. You can also model gas combustion using the finite-rate chemistry model.

You also learned how to set up and solve a gas phase combustion problem using the Discrete Ordinates radiation model, and applying the appropriate absorption coefficient.

## 14.6. References

 A. Sayre, N. Lallement, J. Dugu, and R. Weber "Scaling Characteristics of Aerodynamics and Low-NOx Properties of Industrial Natural Gas Burners", The SCALING 400 Study, Part IV: The 300 KW BERL Test Results, IFRF Doc No F40/y/11, International Flame Research Foundation, The Netherlands.

## 14.7. Further Improvements

This tutorial guides you through the steps to first generate an initial solution, and then to reach a more accurate second-order solution. You may be able to increase the accuracy of the solution even further by using an appropriate higher-order discretization scheme and by adapting the mesh. Mesh adaption can also ensure that your solution is independent of the mesh. These steps are demonstrated in Introduction to Using ANSYS Fluent: Fluid Flow and Heat Transfer in a Mixing Elbow (p. 121).

# **Chapter 15: Modeling Surface Chemistry**

This tutorial is divided into the following sections:

- 15.1. Introduction 15.2. Prerequisites
- 15.3. Problem Description 15.4. Setup and Solution
- 15.4. Setup and So 15.5. Summary
- 15.6. Further Improvements

# 15.1. Introduction

In chemically reacting laminar flows, such as those encountered in chemical vapor deposition (CVD) applications, accurate modeling of time-dependent hydrodynamics, heat and mass transfer, and chemical reactions (including wall surface reactions) is important.

In this tutorial, surface reactions are considered. Modeling the reactions taking place at gas-solid interfaces is complex and involves several elementary physicochemical processes like adsorption of gas-phase species on the surface, chemical reactions occurring on the surface, and desorption of gases from the surface back to the gas phase.

This tutorial demonstrates how to do the following:

- · Create new materials and set the mixture properties.
- Model surface reactions involving site species.
- Enable physical models and define boundary conditions for a chemically reacting laminar flow involving wall surface reactions.
- Calculate the deposition solution using the pressure-based solver.
- Examine the flow results using graphics.

# 15.2. Prerequisites

This tutorial is written with the assumption that you have completed one or more of the introductory tutorials found in this manual:

- Introduction to Using ANSYS Fluent in ANSYS Workbench: Fluid Flow and Heat Transfer in a Mixing Elbow (p. 1)
- Parametric Analysis in ANSYS Workbench Using ANSYS Fluent (p. 73)
- Introduction to Using ANSYS Fluent: Fluid Flow and Heat Transfer in a Mixing Elbow (p. 121)

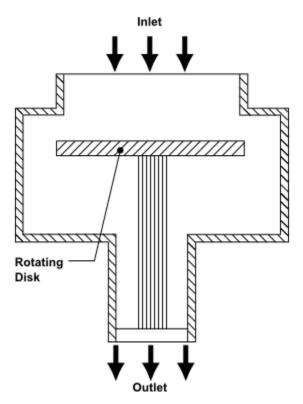
and that you are familiar with the ANSYS Fluent tree and ribbon structure. Some steps in the setup and solution procedure will not be shown explicitly.

Before beginning with this tutorial, see the Fluent User's Guide for more information about species transport, chemically reacting flows, wall surface reaction modeling, and chemical vapor deposition. In particular, you should be familiar with the Arrhenius rate equation, as this equation is used for the surface reactions modeled in this tutorial.

# **15.3. Problem Description**

A rotating disk CVD reactor for the growth of Gallium Arsenide (GaAs) shown in Figure 15.1: Schematic of the Reactor Configuration (p. 618) will be modeled.

Figure 15.1: Schematic of the Reactor Configuration



The process gases, Trimethyl Gallium  $(Ga(CH_3)_3)$  and Arsine  $(AsH_3)$  enter the reactor at 293 K through the inlet at the top. These gases flow over the hot, spinning disk depositing thin layers of gallium and arsenide on it in a uniform, repeatable manner. The disk rotation generates a radially pumping effect, which forces the gases to flow in a laminar manner down to the growth surface, outward across the disk, and finally to be discharged from the reactor.

The semiconductor materials Ga(s) and As(s) are deposited on the heated surface governed by the following surface reactions.

$$AsH_3 + Ga\_s \rightarrow Ga + As\_s + 1.5H_2 \tag{15.1}$$

$$Ga(CH_3)_3 + As_s \rightarrow As + Ga_s + 3CH_3$$
(15.2)

The inlet gas is a mixture of Trimethyl Gallium, which has a mass fraction of 0.15, and Arsine, which has a mass fraction of 0.4, the remainder is hydrogen. The mixture velocity at the inlet is 0.02189 m/s. The disk rotates at 80 rad/sec. The top wall (wall-1) is heated to 473 K and the sidewalls (wall-2) of the reactor are maintained at 343 K. The susceptor (wall-4) is heated to a uniform temperature of 1023 K and the bottom wall (wall-6) is at 303 K. These CVD reactors are typically known as cold-wall reactors, where

only the wafer surface is heated to higher temperatures, while the remaining reactor walls are maintained at low temperatures.

In this tutorial, simultaneous deposition of Ga and As is simulated and examined. The mixture properties and the mass diffusivity are determined based on kinetic theory. Detailed surface reactions with multiple sites and site species, and full multi-component/thermal diffusion effects are also included in the simulation.

The purpose of this tutorial is to demonstrate surface reaction capabilities in ANSYS Fluent. Convective heat transfer is considered to be the dominant mechanism compared to radiative heat transfer, thus radiation effects are ignored.

# 15.4. Setup and Solution

The following sections describe the setup and solution steps for this tutorial:

- 15.4.1. Preparation
- 15.4.2. Reading and Checking the Mesh
- 15.4.3. Solver and Analysis Type
- 15.4.4. Specifying the Models
- 15.4.5. Defining Materials and Properties
- 15.4.6. Specifying Boundary Conditions
- 15.4.7. Setting the Operating Conditions
- 15.4.8. Simulating Non-Reacting Flow
- 15.4.9. Simulating Reacting Flow
- 15.4.10. Postprocessing the Solution Results

## 15.4.1. Preparation

To prepare for running this tutorial:

- 1. Set up a working folder on the computer you will be using.
- 2. Go to the ANSYS Customer Portal, https://support.ansys.com/training.

### Note

If you do not have a login, you can request one by clicking **Customer Registration** on the log in page.

- 3. Enter the name of this tutorial into the search bar.
- 4. Narrow the results by using the filter on the left side of the page.
  - a. Click ANSYS Fluent under Product.
  - b. Click **18.0** under **Version**.
- 5. Select this tutorial from the list.
- 6. Click the **surface\_chem\_R180.zip** link to download the input files.
- 7. Unzip the surface\_chem\_R180.zip file you have downloaded to your working folder.

The file surface.msh can be found in the surface\_chem folder created after unzipping the file.

8. Use Fluent Launcher to start the **3D** version of ANSYS Fluent.

Fluent Launcher displays your **Display Options** preferences from the previous session.

For more information about the Fluent Launcher, see starting ANSYS Fluent using the Fluent Launcher in the Fluent Getting Started Guide.

- 9. Ensure that the **Display Mesh After Reading** and **Workbench Color Scheme** options are enabled.
- 10. Ensure that the Serial processing option is selected.
- 11. Enable **Double Precision**.

## 15.4.2. Reading and Checking the Mesh

1. Read in the mesh file surface.msh.



2. Check the mesh.



ANSYS Fluent will perform various checks on the mesh and will report the progress in the console. Ensure that the reported minimum volume is a positive number.

3. Scale the mesh.



Scale the mesh to meters as it was created in centimeters.

Scale Mesh				
- Domain E	Extents			Scaling
Xmin (m)	-0.2791186	Xmax (m)	0.2791333	Convert Units
Ymin (m)	-0.2793909	Ymax (m)	0.2794	Specify Scaling Factors
Zmin (m)	0	Zmax (m)	0.454	Mesh Was Created In
Zmin (m) 0 Zmax (m) 0.454			Scaling Factors       X     0.01       Y     0.01       Z     0.01       Scale     Unscale	
Close Help				

- a. Select **cm** (centimeters) from the **Mesh Was Created In** drop-down list in the **Scaling** group box.
- b. Click Scale and verify that the domain extents are as shown in the Scale Mesh dialog box.

The default SI units will be used in this tutorial, hence there is no need to change any units.

- c. Close the Scale Mesh dialog box.
- 4. Check the mesh.

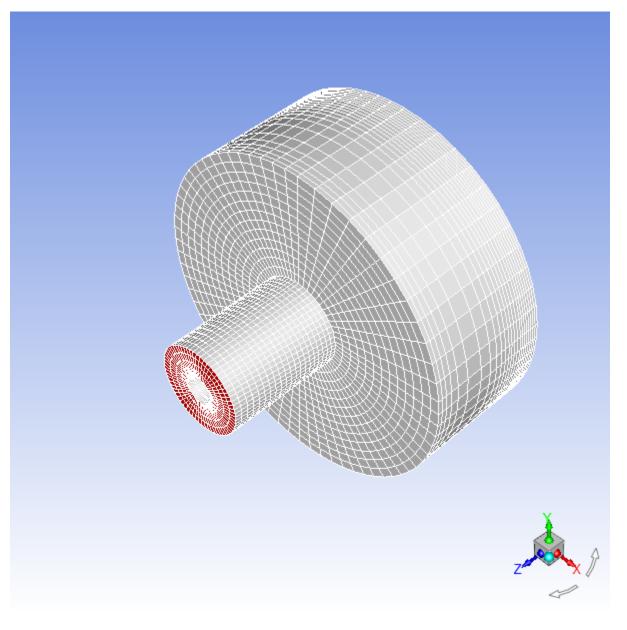
Setting Up Domain  $\rightarrow$  Mesh  $\rightarrow$  Check

#### Note

It is a good practice to check the mesh after manipulating it (scale, convert to polyhedra, merge, separate, fuse, add zones, or smooth and swap). This will ensure that the quality of the mesh has not been compromised.

5. Examine the mesh (Figure 15.2: Mesh Display (p. 622)).

### Figure 15.2: Mesh Display



### Extra

You can use the left mouse button to rotate the image and view it from different angles. Use the right mouse button to check which zone number corresponds to each boundary. If you click the right mouse button on one of the boundaries in the graphics window, its name and type will be printed in the ANSYS Fluent console. This feature is especially useful when you have several zones of the same type and you want to distinguish between them quickly. Use the middle mouse button to zoom the image.

## 15.4.3. Solver and Analysis Type

Retain the default solver settings of pressure-based steady-state solver in the **Setting Up Physics** tab (**Solver** group).

Setting Up Physics → Solver				
		Solver		
Time Steady	Type Pressure-Based	Velocity Formulation Absolute	Operating Conditions	
<ul> <li>Transient</li> </ul>	<ul> <li>Density-Based</li> </ul>	<ul> <li>Relative</li> </ul>	Reference Values	

## 15.4.4. Specifying the Models

In this problem, the energy equation and the species conservation equations will be solved, along with the momentum and continuity equations.

1. Enable heat transfer by turning on the energy equation.

**Setting Up Physics**  $\rightarrow$  **Models**  $\rightarrow$  **Energy** 

		Models	
	Radiation	Nultiphase	🚨 Solidify/Melt
Energy	∦ <sub>‡</sub> Heat Exchanger	♂ Species	动) Acoustics
	🔄 Viscous	📑 Discrete Phase	🗄 More

2. Enable chemical species transport.

10001	10001 36	Setting	Up Phy	ysics $\rightarrow$	Models	$\rightarrow$	Species
-------	----------	---------	--------	---------------------	--------	---------------	---------

Species Model	<b>—</b>
Model Off Species Transport Non-Premixed Combustion Premixed Combustion Partially Premixed Combustion Composition PDF Transport	Mixture Properties Mixture Material mixture-template View Import CHEMKIN Mechanism Number of Volumetric Species 3
Reactions	
Options <ul> <li>Inlet Diffusion</li> <li>Diffusion Energy Source</li> <li>Full Multicomponent Diffusion</li> <li>Thermal Diffusion</li> </ul>	Select Boundary Species Select Monitored Species
ОК Ар	ply Cancel Help

a. Select Species Transport in the Model list.

The **Species Model** dialog box will expand to show relevant input options.

b. Retain the selection of mixture-template from the Mixture Material drop-down list.

You will modify the mixture material later in this tutorial.

c. Retain the default setting for **Diffusion Energy Source**.

This includes the effect of enthalpy transport due to species diffusion in the energy equation, which contributes to the energy balance, especially for the case of Lewis numbers far from unity.

d. Enable Full Multicomponent Diffusion and Thermal Diffusion.

The **Full Multicomponent Diffusion** activates Stefan-Maxwell's equations and computes the diffusive fluxes of all species in the mixture to all concentration gradients. The **Thermal Diffusion** effects cause heavy molecules to diffuse less rapidly, and light molecules to diffuse more rapidly, toward heated surfaces.

e. Click OK to close the Species Model dialog box.

## **15.4.5. Defining Materials and Properties**

In the following steps you will copy the gas-phase species  $(AsH_3, Ga(CH_3)_3, CH_3, and H_2)$  from the ANSYS Fluent database, specify the mixture materials, setup the reactions, and modify the material properties. You will also create the site species (Ga\_s and As\_s) and the solid species (Ga and As).

1. Copy arsenic-trihydride, hydrogen, methyl-radical, and trimethyl-gallium from the ANSYS Fluent material database to the list of fluid materials and modify their properties.



- a. Click Fluent Database... in the Create/Edit Materials dialog box to open the Fluent Database Materials dialog box.
- b. In the Fluent Database Materials dialog box, select fluid from the Material Type drop-down list.
- c. From the Fluent Fluid Materials selection list, select arsenic-trihydride (ash3), hydrogen (h2), methylradical (ch3), and trimethyl-gallium (game3) by clicking each species once.

Scroll down the Fluent Fluid Materials list to locate each species.

Fluent Database Materials			
Fluent Fluid Materials [4/563]	= =	Material Type	•
trimethyl-aluminum-dimer (al2me6) trimethyl-arsenic (asme3) trimethyl-chloro-silane (clsi <ch3>3)</ch3>		<ul> <li>Order Mate</li> <li>Name</li> <li>Chemica</li> </ul>	rials by
trimethyl-gallium (game3) trimethyl-gallium-arsenic-trihydride (h trimethyl-methylene-silane (si <ch3>3c</ch3>	-	•	
Copy Materials from Case Delete			
Properties			
Cp (Specific Heat) (j/kg-k)	piecewise-polynomial		View
Molecular Weight (kg/kmol)			▼ View
	77.94551		
Standard State Enthalpy (j/kgmol)	constant		▼ View
	6.956396e+07		
Standard State Entropy (j/kgmol-k)	constant		▼ View
	232871.2		
New) Edit	Save Copy	Close Help	h.

- d. Click **Copy** to copy the selected species to your model.
- e. Click Close to close the Fluent Database Materials dialog box.

The **Create/Edit Materials** dialog box is updated to show the new materials, **arsenic-trihydride** (ash3), hydrogen (h2), methyl-radical (ch3), and trimethyl-gallium (game3), in the Fluent Fluid Materials drop-down list. The species are also listed under Fluid in the **Materials** task page and the **Setup/Materials/Fluid** tree branch.

- 2. Create the site species (Ga\_s and As\_s) and the solid species (Ga and As).
  - a. In the Create/Edit Materials dialog box, select air from the Fluent Fluid Materials drop-down list.
  - b. Enter ga\_s for the **Name** text entry field.
  - c. Enter ga\_s for the Chemical Formula text entry field.
  - d. Click **Change/Create** to create the new material.
  - e. Click No in the Question dialog box when asked if you want to overwrite air.

The new material **ga\_s** is added to your model and listed under **Fluid** in the **Materials** task page and under the **Setup/Materials/Fluid** tree branch.

f. Create As\_s, Ga, and As following the same procedure as for Ga\_s and close the **Create/Edit Materials** dialog box.

### Extra

To enter complex formulae such as  $Ga(CH_3)_3$  in the text entry box, use '<' and '>' instead of '(' and ')', respectively.

3. Set the mixture species.

Setup  $\rightarrow$  Materials  $\rightarrow$  Mixture  $\rightarrow$  mixture-template  $\stackrel{\frown}{\rightarrow}$  Edit...

- a. Enter gaas\_deposition for Name.
- b. Click Change/Create.
- c. Click Yes in the Question dialog box to overwrite the mixture-template.
- d. Set the Selected Species.
  - i. In **Properties** group box, click the **Edit...** button to the right of the **names** drop-down list for **Mixture Species** to open the **Species** dialog box.

Species	<b>—</b>
Mixture gaas_deposition	
Available Materials	Selected Species
ga     ▲       as_s     ga_s       air     ■       nitrogen (n2)     ∞       oxygen (o2)     ■       water-vapor (h2o)     ▼	ash3 game3 ch3 h2 Add Remove
Selected Site Species	Selected Solid Species
Add Remove	Add Remove
OK Can	cel Help

ii. Set the **Selected Species** from the **Available Materials** selection list as shown in Table 15.1: Selected Species (p. 627).

Table	15.1:	Selected	<b>Species</b>
-------	-------	----------	----------------

	Selected Species
ash3	
game3	
ch3	
h2	

#### Important

- Add arsenic-trihydride (ash3), trimethyl-gallium (game3), methyl-radical (ch3), and hydrogen (h2) to the Selected Species list before removing h2o, o2, and n2.
- Ensure that **h2** is at the bottom in the **Selected Species** selection list as shown in Table 15.1: Selected Species (p. 627). ANSYS Fluent will interpret the last species in the list as the bulk species.

#### Note

To add/remove the species:

- To add a particular species to the list, select the required species from the Available Materials selection list and click Add in the corresponding species selection list (Selected Species, Selected Site Species, or Selected Solid Species). The species will be added to the end of the relevant list and removed from the Available Materials list.
- To remove an unwanted species from the selection list, select the species from the selection list (Selected Species, Selected Site Species, or Selected Solid Species) and click Remove in the corresponding selection list. The species will be removed from the list and added to the Available Materials list.
- iii. Click **OK** to close the **Species** dialog box.
- iv. Click Change/Create and close the Creat/Edit Materials dialog box.
- 4. Enable chemical species transport reaction.

Setting Up Physics  $\rightarrow$  Models  $\rightarrow$  Species...

Although you enable reactions, you still run a non-reacting flow to produce an initial solution. You will run a reacting flow in Simulating Reacting Flow (p. 644).

Species Model	
Model <ul> <li>Off</li> <li>Species Transport</li> <li>Non-Premixed Combustion</li> <li>Premixed Combustion</li> <li>Partially Premixed Combustion</li> </ul>	Mixture Properties Mixture Material gaas_deposition  Edit Import CHEMKIN Mechanism Number of Volumetric Species 4
Composition PDF Transport	Number of Solid Species 0 Number of Site Species 0
<ul> <li>Volumetric</li> <li>Wall Surface</li> <li>Particle Surface</li> <li>Electrochemical</li> <li>Wall Surface Reaction Options</li> <li>Heat of Surface Reactions</li> <li>Mass Deposition Source</li> <li>Aggressiveness Factor 0</li> <li>Chemistry Solver</li> <li>None - Explicit Source</li> </ul>	Turbulence-Chemistry Interaction <ul> <li>Finite-Rate/No TCI</li> <li>Finite-Rate/Eddy-Dissipation</li> <li>Eddy-Dissipation</li> <li>Eddy-Dissipation Concept</li> </ul> Coal Calculator
Options <ul> <li>Inlet Diffusion</li> <li>Diffusion Energy Source</li> <li>Full Multicomponent Diffusion</li> <li>Thermal Diffusion</li> </ul>	Select Boundary Species Select Monitored Species
ОК Арріу	Cancel Help

- a. Enable Volumetric and Wall Surface in the Reactions group box.
- b. Retain the selection of gass\_deposition from the Mixture Material drop-down list.
- c. Disable Heat of Surface Reactions and enable Mass Deposition Source.
- d. Click **OK** to close the **Species Model** dialog box.
- 5. Set the site and solid species and the mixture reactions in a similar manner to the mixture species.



a. Click the **Edit...** button to the right of the **names** drop-down list for **Mixture Species** in the **Properties** group box.

Specify the **Selected Site Species** and the **Selected Solid Species** as shown in Table 15.2: Selected Site and Solid Species (p. 629).

Table 15.2: Selected Site and Solid Species

Selected Site Species	Selected Solid Species	
ga_s	ga	
as_s	as	

Once you set the site and solid species, the **Species** dialog box should look like this:

Species	<b>—</b>
Mixture gaas_deposition	
Available Materials	Selected Species
water-vapor (h2o) oxygen (o2) nitrogen (n2) air	ash3 game3 ch3 h2 Add Remove
Selected Site Species	Selected Solid Species
ga_s as_s	ga as
Add Remove	Add Remove
OK Can	cel Help

- b. Click **OK** to close the **Species** dialog box.
- c. Click the Edit... button to the right of the Reaction drop-down list to open the Reactions dialog box.

Reactions						<b>X</b>
Mixture gaas_deposition			Total Nur	nber of Reactions	2 🌲	
Reaction Name     ID     Reaction Type       gallium-dep     1     Image: Comparison of the state o						
Number of Reactants 2			Number of Produ	icts 3 🌻		
Species	Stoich. Coefficient	Rate Exponent	Species	Stoich. Coefficient	Rate Exponent	
ash3 🔻	1	1	ga	▼ 1	0	≡
[ga_s ▼	1	1	as_s	• 1	0	
			h2	▼ 1.5	0	
Arrhenius Rate			Mixing Rate			
Pre-Exponential Fac	tor 1000000		A 4	B 0.5		
Activation Energy (j/kgm	o (lor					
Temperature Expone	ent 0.5					
Include Backward Rea	action S	pecify				
Third-Body Efficiencie	s S	pecify				
Pressure-Dependent		pecify				
Coverage-Dependent Reaction Specify						
		OK Ca	ancel Help			H

d. Increase the **Total Number of Reactions** to 2, and define the following reactions using the parameters in Table 15.3: Reaction Parameters (p. 630) :

$$AsH_3 + Ga\_s \rightarrow Ga + As\_s + 1.5H_2 \tag{15.3}$$

$$Ga(CH_3)_3 + As_s \rightarrow As + Ga_s + 3CH_3$$
(15.4)

### Table 15.3: Reaction Parameters

Parameter	For Equation 15.3 (p. 630)	For Equation 15.4 (p. 630)
Reaction ID	1	2 <sup>a</sup>
Reaction Name	gallium-dep	arsenic-dep
Reaction Type	Wall Surface	Wall Surface
Number of Reactants	2	2
Species	ash3, ga_s	game3, as_s
Stoich. Coefficient	ash3= 1, ga_s= 1	game3= 1, as_s= 1
Rate Exponent	ash3= 1, ga_s= 1	game3= 1, as_s= 1
Arrhenius Rate	PEF= 1e+06, AE= 0, TE= 0.5 <sup>b</sup>	<b>PEF=</b> 1e+12, <b>AE=</b> 0, <b>TE=</b> 0.5

Parameter	For Equation 15.3 (p. 630)	For Equation 15.4 (p. 630)
Number of Products	3	3
Species	ga, as_s, h2	as, ga_s, ch3
Stoich. Coefficient	ga= 1, as_s= 1, h2= 1.5	as= 1, ga_s= 1, ch3= 3
Rate Exponent	<b>as_s=</b> 0, <b>h2</b> = 0	<b>ga_s=</b> 0, <b>ch3=</b> 0

<sup>a</sup>Set the  $\mathbf{ID}$  to 2 in order to set the parameters for the second reaction.

<sup>b</sup>Here, PEF = **Pre-Exponential Factor**, AE = **Activation Energy**, and TE = **Temperature Exponent**.

- e. Click OK to save the data and close the Reactions dialog box.
- f. Set the reaction mechanisms for the mixture.
  - i. Click the **Edit...** button to the right of the **Mechanism** drop-down list to open the **Reaction Mechanisms** dialog box.

Reaction Mechanisms			×						
Number of Mechanisms 1 🚔 Mechanism I	Number of Mechanisms 1 🚔 Mechanism ID 1 🚔 Name gaas-ald								
Reaction Type Volumetric Vall Surface I All									
Reactions [2/2]	Number of Sites 1 🚔								
gallium-dep	Site Name	Site Density [kgmol/m2]							
arsenic-dep	site-1	1e-08	Define						
	site-2	0	Define						
	site-3	0	Define						
	OK Cancel Help								

- ii. Retain Number of Mechanisms as 1.
- iii. Enter gaas-ald for Name.
- iv. Select Wall Surface in the Reaction Type group box.
- v. Select gallium-dep and arsenic-dep from the Reactions selection list.
- vi. Set Number of Sites to 1.
- vii. Enter 1e-08 kgmol/m<sup>2</sup> for **Site Density** for **site-1**.

viii.Click the **Define...** button to the right of site-1 to open the **Site Parameters** dialog box.

	Site Parameters	<b>—</b>							
Site	Site Name site-1								
Tot	al Number of Site Species 2								
Sit	e Species	Initial Site Coverage 🔺							
ga	a_s 🔻	• 0.7							
as	s_s •	• 0.3							
ga	a_s 🔻	- 0							
ga	a_s 🔻								
	Apply Close	Help							

- A. Set Total Number of Site Species to 2.
- B. Select **ga\_s** as the first site species and enter 0.7 for **Initial Site Coverage**.
- C. Select **as\_s** as the second site species and enter 0.3 for **Initial Site Coverage**.
- D. Click **Apply** and close the **Site Parameters** dialog box.
- ix. Click **OK** to close the **Reaction Mechanisms** dialog box.
- g. Retain the default selection of incompressible-ideal-gas from the Density drop-down list.
- h. Retain the default selection of mixing-law from the Cp (Specific Heat) drop-down list.
- i. Select mass-weighted-mixing-law from the Thermal Conductivity drop-down list.
- j. Select mass-weighted-mixing-law from the Viscosity drop-down list.
- k. Retain the default selection of kinetic-theory from the Mass Diffusivity drop-down list.
- I. Retain the default selection of **kinetic-theory** from the **Thermal Diffusion Coefficient** drop-down list.
- m. Click Change/Create and close the Create/Edit Materials dialog box.
- 6. Specify the material properties for arsenic-trihydride, hydrogen, methyl-radical, trimethyl-gallium, site species (Ga\_s and As\_s), and solid species (Ga and As).

Setup  $\rightarrow$  Materials  $\rightarrow$  Mixture  $\rightarrow$  gaas\_deposition  $\rightarrow$  arsenic-trihydride  $\stackrel{\bigcirc}{\Box}$  Edit...

Create/Edit Materials				×
Vame arsenic-trihydride Chemical Formula ash3		Material Type fluid Fluent Fluid Materials arsenic-trihydride (ash3) Mixture gaas_deposition	•	Order Materials by <ul> <li>Name</li> <li>Chemical Formula</li> </ul> Fluent Database User-Defined Database
Properties		gaas_ucposition		
Standard State Entropy (j/kgmol-k)	constant	▼ Edit	*	
	130579.1			
Reference Temperature (k)	constant	▼ Edit		
	298.15			
L-J Characteristic Length (angstrom)	constant	▼ Edit		
	4.145			
L-J Energy Parameter (k)	constant	▼ Edit	E	
	259.8			
			Ŧ	
Chang	e/Create	Delete Close Help		

a. In the **Properties** group box, modify the arsenic-trihydride properties as shown in Table 15.4: Properties of Species (p. 633).

#### Important

Ensure Mixture is set to gaas\_deposition

#### Tip

Scroll down in the **Properties** group box to see all the parameters.

Table 15.4:	Properties	of Species
-------------	------------	------------

Parameter	AsH_3	Ga(CH_3)_3	CH_3	H_2
Name	arsenic-tri- hydride	trimethyl- gallium	methyl-radic- al	hydrogen
Chemical Formula	ash3	game3	ch3	h2
Cp (Specific Heat)	piecewise- polynomial	piecewise-poly- nomial	piecewise-poly- nomial	piecewise-poly- nomial
Thermal Conductiv- ity	kinetic-theory	kinetic-theory	kinetic-theory	kinetic-theory

Parameter	AsH_3	Ga(CH_3)_3	CH_3	H_2
Viscosity	kinetic-theory	kinetic-theory	kinetic-theory	kinetic-theory
Molecular Weight	77.95	114.83	15	2.02
Standard State En- thalpy	0	0	2.044e+07	0
Standard State En- tropy	130579.1	130579.1	257367.6	130579.1
Reference Temper- ature	298.15	298.15	298.15	298.15
L-J Characteristic Length	4.145	5.68	3.758	2.827
L-J Energy Paramet- er	259.8	398	148.6	59.7

b. When finished, click **Change/Create** to update your local copy of the species material.

#### Note

When you modify the properties of the material local copy, the original copy in Fluent material database stays intact.

c. In a similar way, modify the properties of trimethyl-gallium (game3), methyl-radical (ch3), and hydrogen (h2).

#### Note

Make sure to click **Change/Create** each time you modify the properties for the material to apply the changes to the local copy.

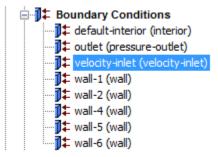
- d. Select ga\_s from the Fluent Fluid Materials drop-down list.
- e. Enter the parameter values for the ga\_s species as shown in Table 15.5: Properties of Species (p. 634).

Parameter	Ga_s	As_s	Ga	As
Name	ga_s	as_s	ga	as
Chemical Formula	ga_s	as_s	ga	as
Cp (Specific Heat)	520.64	520.64	1006.43	1006.43
Thermal Conduct- ivity	0.0158	0.0158	kinetic-theory	kinetic-theory
Viscosity	2.125e-05	2.125e-05	kinetic-theory	kinetic-theory
Molecular Weight	69.72	74.92	69.72	74.92

Parameter	Ga_s	As_s	Ga	As
Standard State Enthalpy	-3117.71	-3117.71	0	0
Standard State Entropy	154719.3	154719.3	0	0
Reference Temper- ature	298.15	298.15	298.15	298.15
L-J Characteristic Length	0	0	0	0
L-J Energy Para- meter	0	0	0	0

- f. Modify the material properties for As\_s, Ga, and As as shown in Table 15.5: Properties of Species (p. 634).
- g. Close the **Create/Edit Materials** dialog box.

## **15.4.6. Specifying Boundary Conditions**



1. Set the conditions for **velocity-inlet**.

Velocity Inlet							×
Zone Name velocity-inlet							
Momentum	Thermal	Radiation	Species	DPM	Multiphase	Potential	UDS
Velocit	y Specificatio	on Method M	agnitude, N	ormal to	Boundary		•
	Refere	nce Frame A	bsolute				•
	Velocity Ma	gnitude (m/s	) 0.02189		const	tant	•
Supersonic/Init	ial Gauge Pre	ssure (pascal	) 0		cons	tant	•
		0	K Cancel	Help			

**F**Setup  $\rightarrow$  Boundary Conditions  $\rightarrow$  velocity-inlet  $\stackrel{0}{\hookrightarrow}$  Edit...

- a. Retain the default selection of **Magnitude**, Normal to Boundary from the Velocity Specification Method drop-down list.
- b. Retain the default selection of **Absolute** from the **Reference Frame** drop-down list.
- c. Enter 0.02189 m/s for Velocity Magnitude.
- d. Click the **Thermal** tab and enter 293 K for **Temperature**.
- e. Under the Species tab, set the Species Mass Fractions for ash3 to 0.4, game3 to 0.15, and ch3 to 0.

The mass fraction of hydrogen is 0.45, but there is no need to specify this since it is the last species in the mixture.

💶 Velocity Inl	let						<b>—</b> ×
Zone Name							
velocity-inlet							
Momentum	Thermal	Radiation	Species	DPM	Multiphase	Potential	UDS
[	Specify Spec	cies in Mole F	ractions				
Species Mass	Fractions						
ash3	0.4		constant		•		
game3	0.15		constant		•		
ch3	0		constant		•		
	OK Cancel Help						

- f. Click **OK** to close the **Velocity Inlet** dialog box.
- 2. Set the boundary conditions for **outlet**.



a. Retain the default settings under the Momentum tab.

Pressure Out	let						×
Zone Name							
outlet							
Momentum	Thermal	Radiation	Species	DPM	Multiphase	Potential	UDS
В	ackflow Refe	rence Frame	Absolute				•
	Gauge F	Pressure (paso	al) 0		cor	nstant	•
Backflow Direc	tion Specifica	ition Method	Normal to B	oundary			•
Backf	ow Pressure	Specification	Total Press	ire			•
🔲 Radial Equili	Radial Equilibrium Pressure Distribution						
🔲 Average Pr	essure Specif	ication					
Target Mass Flow Rate							
OK Cancel Help							

- b. Under the Thermal tab, enter 400 K for **Temperature**.
- c. Under the Species tab, set the Species Mass Fractions for ash3 to 0.32, game3 to 0.018, and ch3 to 0.06.

Since a certain amount of backflow is expected in the flow regions around the rotating shaft, you should set the realistic backflow species mass fractions to minimize convergence difficulties.

Pressure O	utlet					
Zone Name						
outlet						
Momentum	Thermal Radiation	n <b>Species</b> DPM Multiphase Potential UDS				
	Specify Species in Mo	ole Fractions				
Species Mass	Fractions					
ash3	0.32	constant				
game3	0.018	constant 🔹				
ch3	ch3 0.06 constant 💌					
OK Cancel Help						

d. Click **OK** to accept the remaining default settings.

3. Set the boundary conditions for **wall-1**.



a. Click the **Thermal** tab.

🖬 Wall							<b>—</b> ×—
Zone Name							
wall-1							
Adjacent Cell Zone							
fluid							
Momentum Thermal	Radiation Species	DPM	Multiphase	UDS	Wall Film	Potential	
Thermal Conditions							
Heat Flux		Temperature	e (k) 473		cons	stant	•
Temperature			Wall	Thickness	(m) 0		P
Convection	Heat Generati	on Rate (w/	m3) 0		cons	stant	•
<ul> <li>Radiation</li> <li>Mixed</li> </ul>				I Conductio			Edit
<ul> <li>Mixed</li> <li>via System Coupling</li> </ul>				Conductor	1 Layer		Cultin
<ul> <li>via Mapped Interface</li> </ul>							
Material Name							
aluminum	Edit						
Lucianteri							
	(	OK Canc	el Help				.4

- i. Select Temperature in the Thermal Conditions group box.
- ii. Enter 473 K for **Temperature**.
- b. Click **OK** to close the **Wall** dialog box.
- 4. Set the boundary conditions for **wall-2**.

**E** Setup  $\rightarrow$  Boundary Conditions  $\rightarrow$  wall-2  $\stackrel{0}{\hookrightarrow}$  Edit...

- a. Click the Thermal tab.
  - i. Select Temperature in the Thermal Conditions group box.
  - ii. Enter 343 K for Temperature.
- b. Click **OK** to close the **Wall** dialog box.
- 5. Set the boundary conditions for **wall-4**.

**E** Setup  $\rightarrow$  Boundary Conditions  $\rightarrow$  wall-4  $\stackrel{\text{D}}{\hookrightarrow}$  Edit...

Wall Zone Name wall-4 Adjacent Cell Zone	A	_	_						×
fluid									
Momentum	Thermal	Radiation	Species	DPM	Multiphase	UDS	Wall Film	Potential	
Wall Motion Stationary V Moving Wall	U .		acent Cell Zone		ed (rad/s) 80 on-Axis Origin			nstant Axis Direction	¥
	0	Translational Rotational		X (m) Y (m)	0	P	X 0 Y 0	ALS DIECOUN	P
	0	Components		Z (m)	Z (m) 0		Ζ1		P
Shear Condition No Sip Specified Sh Specularity ( Marangoni S	ear Coefficient								
Wall Roughness	s								
Roughness Heig	ht (m) 0		consta	nt	Ψ				
Roughness Co	nstant 0.5		consta	nt	-				
			(	OK Cano	el Help				

a. Select Moving Wall in the Wall Motion group box.

The **Wall** dialog box will expand to wall motion inputs and options.

- b. Select Absolute and Rotational in the Motion group box.
- c. Enter 80 rad/s for Speed.
- d. Retain the other default settings.
- e. Click the Thermal tab.
  - i. Select Temperature in the Thermal Conditions group box.
  - ii. Enter 1023 K for Temperature.
- f. Click the **Species** tab.

💶 Wall				<b>—</b> ×	
Zone Name		_			
wall-4					
Adjacent Cell Zone		_			
fluid					
Momentum Thermal Radiation	Species DPM	Multiphase	UDS Wall Film	Potential	
✓ Reaction Reaction Mechanism gaas-ald Surface Area Washcoat Factor 1	F				
OK Cancel Heb					

- i. Enable Reaction.
- ii. Retain the selection of gaas-ald from the Reaction Mechanisms drop-down list.
- g. Click **OK** to close the **Wall** dialog box.
- 6. Set the boundary conditions for wall-5.

**E** Setup  $\rightarrow$  Boundary Conditions  $\rightarrow$  wall-5  $\stackrel{0}{\hookrightarrow}$  Edit...

- a. Select Moving Wall in the Wall Motion group box.
- b. Select **Absolute** and **Rotational** in the **Motion** group box.
- c. Enter 80 rad/s for Speed.
- d. Click the Thermal tab.
  - i. Select Temperature in the Thermal Conditions group box.
  - ii. Enter 720 K for Temperature.
- e. Click **OK** to close the **Wall** dialog box.
- 7. Set the boundary conditions for **wall-6**.

**E** Setup  $\rightarrow$  Boundary Conditions  $\rightarrow$  wall-6  $\stackrel{\mu}{\hookrightarrow}$  Edit...

- a. Click the **Thermal** tab.
  - i. Select Temperature in the Thermal Conditions group box.
  - ii. Enter 303 K for Temperature.
- b. Click **OK** to close the **Wall** dialog box.

## 15.4.7. Setting the Operating Conditions

1. Setting Up Physics → Solver → Operating Conditions...

Solver						
Time Steady	Type Pressure-Based	Velocity Formulation Output	Operating Conditions			
<ul> <li>Transient</li> </ul>		<ul> <li>Relative</li> </ul>	🥏 Reference Values			

Operating Conditions	<b>X</b>			
Pressure Operating Pressure (pascal) 10000 P Reference Pressure Location X (m) 0 P Y (m) 0 P Z (m) 0 P	Gravity Gravity Gravitational Acceleration X (m/s2) 0 Y (m/s2) 0 Z (m/s2) 9.81 Boussinesq Parameters Operating Temperature (k) 303 P Variable-Density Parameters Specified Operating Density			
OK Cancel Help				

- a. Enter 10000 Pa for **Operating Pressure**.
- b. Enable Gravity.

The dialog box will expand to show related gravitational inputs.

- c. Enter  $9.81 \text{ m/s}^2$  for **Gravitational Acceleration** in the **Z** direction.
- d. Enter 303 K for **Operating Temperature**.
- e. Click **OK** to close the **Operating Conditions** dialog box.

The **Operating Conditions** dialog box can be accessed from the **Cell Zone Conditions** task page.

# 15.4.8. Simulating Non-Reacting Flow

1. Disable **Volumetric** for solving non-reacting flow.

**Setting Up Physics**  $\rightarrow$  Models  $\rightarrow$  Species...

- a. Disable **Volumetric** in the **Reactions** group box.
- b. Click **OK** to close the **Species Model** dialog box.

You will first run a non-reacting solution to establish the flow.

2. Select the **Coupled** solver method.

<b>E</b> Solution $\rightarrow$ $\diamondsuit$	Solution	Methods
--	----------	---------

Solution Methods	
Pressure-Velocity Coupling	
Scheme	
Coupled 👻	
Spatial Discretization	
Gradient	*
Least Squares Cell Based	
Pressure	
Second Order	_
Momentum	=
Second Order Upwind 👻	
ash3	
Second Order Upwind 🔹	
game3	
Second Order Upwind 👻	_
Transient Formulation	Ť
Non-Iterative Time Advancement	
Frozen Flux Formulation	
Pseudo Transient	
Warped-Face Gradient Correction	
High Order Term Relaxation Options	
Set All Species Discretizations Together	
Default	
Help	

- a. Select **Coupled** from the **Scheme** drop-down list in the **Pressure-Velocity Coupling** group box.
- b. Retain the default selections in the **Spatial Discretization** group box.

3. Examine **Solution Controls** and retain the default settings.

# **E** Solution $\rightarrow \diamondsuit$ Solution Controls

Solution Controls					
Flow Courant Number					
200					
Explicit Relaxation Factors					
Momentum 0.5					
Pressure 0.5					
Under-Relaxation Factors					
Density	^				
1					
Body Forces					
1					
ash3	Ξ				
1					
game3					
1					
dh3					
1					
	Ŧ				
Default					
Equations Limits Advanced					
Set All Species URFs Together					
Help					

4. Enable residual plotting during the calculation.

**Solving**  $\rightarrow$  **Reports**  $\rightarrow$  **Residuals...** 

- a. Retain the default settings and close the **Residual Monitors** dialog box.
- 5. Initialize the flow field.

**E** Solution  $\rightarrow$  **\clubsuit** Solution Initialization

Solution Initialization
Initialization Methods
<ul> <li>Hybrid Initialization</li> <li>Standard Initialization</li> </ul>
More Settings Initialize
Patch
Reset DPM Sources Reset Statistics
Help

- a. Retain the default selection of **Hybrid Initialization** from the **Initialization Methods** group box.
- b. Click Initialize.
- 6. Save the case file (surface-non-react.cas.gz).



7. Start the calculation by requesting 200 iterations.



a. Enter 200 for No. of Iterations and click Calculate.

The solution will converge in approximately 50 iterations.

### **15.4.9. Simulating Reacting Flow**

1. Enable **Volumetric** for the reacting flow solution.

Setting Up Physics  $\rightarrow$  Models  $\rightarrow$  Species...

Species Model				
Model Off Species Transport Non-Premixed Combustion Premixed Combustion Partially Premixed Combustion Composition PDF Transport Reactions Volumetric Volumetric Particle Surface Particle Surface	Mixture Properties Mixture Material gaas_deposition Edit Import CHEMKIN Mechanism Number of Volumetric Species 4 Number of Solid Species 2 Number of Site Species 2 Turbulence-Chemistry Interaction Finite-Rate/No TCI Finite-Rate/Eddy-Dissipation			
<ul> <li>Electrochemical</li> <li>Wall Surface Reaction Options</li> <li>Heat of Surface Reactions</li> <li>Mass Deposition Source</li> <li>Aggressiveness Factor 0</li> <li>Chemistry Solver</li> <li>None - Explicit Source</li> </ul>	<ul> <li>Eddy-Dissipation</li> <li>Eddy-Dissipation Concept</li> <li>Coal Calculator</li> </ul>			
Options <ul> <li>Inlet Diffusion</li> <li>Diffusion Energy Source</li> <li>Full Multicomponent Diffusion</li> <li>Thermal Diffusion</li> </ul>	Select Boundary Species Select Monitored Species			
OK Apply Cancel Help				

- a. Enable Volumetric and Wall Surface in the Reactions group box.
- b. Ensure that Mass Deposition Source is enabled in the Wall Surface Reaction Options group box.
- c. Click **OK** to close the **Species Model** dialog box.
- 2. Retain the default convergence criteria for calculation.



Residual Monitors					×
Options	Equations				
Print to Console	Residual	Monitor	Check Convergen	nce Absolute Criteria	<u> </u>
V Plot	continuity	$\checkmark$	<b>V</b>	0.001	
Window	x-velocity	<b>V</b>		0.001	
1 🗧 Curves Axes	y-velocity	$\checkmark$		0.001	
Iterations to Plot	z-velocity	$\checkmark$		0.001	
1000 ≑				+	
1000	Residual Values		9	Convergence Criterion	
	Normalize		Iterations	absolute	-
Iterations to Store			5		
1000 🜩	Scale		(	Convergence Conditi	ons
Compute Local Scale					
OK Plot Renormalize Cancel Help					

3. Request 200 more iterations.

**Solving**  $\rightarrow$  Run Calculation  $\rightarrow$  Calculate

The solution will converge in approximately 60 additional iterations.

4. Compute the mass fluxes.

Pos	stprocessing $\rightarrow$	Reports →	Fluxes
103	reprocessing -	Reports	ITUAC

E Flux Reports				
Options	Boundaries Filter Text	Results		
Mass Flow Rate				
Total Heat Transfer Rate	default-interior			
Total Sensible Heat Transfer Rate	outlet	-4.658714721357403e-05		
Radiation Heat Transfer Rate	velocity-inlet	5.5962763594196e-05		
	wall-1			
	wall-2			
	wall-4	-9.373103607053655e-06		
	wall-5			
	wall-6			
	٠	< F		
Save Output Parameter	Save Output Parameter Net Results (kg/s)			
		2.512774e-09		
Compute Write Close Help				

Release 18.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- a. Retain the default selection of Mass Flow Rate in the Options group box.
- b. Select outlet, velocity-inlet, and wall-4 from the Boundaries selection list.

In order to properly assess the mass balance, you must account for the mass deposition on the spinning disk. Hence you select wall-4 in addition to the inlet and outlet boundaries.

c. Click **Compute**, examine the values displayed in the **Results** and **Net Results** boxes, and close the **Flux Reports** dialog box.

The net mass imbalance should be a small fraction (for example, 0.5% or less) of the total flux through the system. If a significant imbalance occurs, you should decrease your residual tolerances by at least an order of magnitude and continue iterating.

5. Display contours of surface deposition rate of **ga** (Figure 15.3: Contours of Surface Deposition Rate of Ga (p. 648)).

Options Filled	Contours of Species
Node Values	Surface Deposition Rate of ga 🔹
<ul> <li>Global Range</li> <li>Auto Range</li> </ul>	Min (kg/m2-s) Max (kg/m2-s)
Clip to Range	0 3.481749e-05
Draw Profiles Draw Mesh	Surfaces Filter Text
Coloring Banded Smooth	default-interior <ul> <li>outlet</li> <li>velocity-inlet</li> <li>wall-1</li> <li>wall-2</li> </ul>
Levels Setup	wall-4 wall-5
20 🜩 1 🌩	New Surface
	Display Compute Close Help

### Postprocessing $\rightarrow$ Graphics $\rightarrow$ Contours $\rightarrow$ Edit...

- a. Enable **Filled** in the **Options** group box.
- b. Select Species... and Surface Deposition Rate of ga from the Contours of drop-down lists.
- c. Select wall-4 from the Surfaces selection list.
- d. Click **Display** and close the **Contours** dialog box.

Rotate the display with the mouse to obtain the view as shown in (Figure 15.3: Contours of Surface Deposition Rate of Ga (p. 648)).

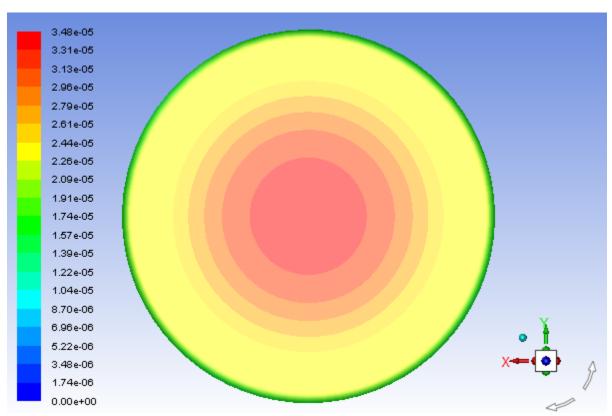


Figure 15.3: Contours of Surface Deposition Rate of Ga

6. Reduce the convergence criteria.

**Solving**  $\rightarrow$  Reports  $\rightarrow$  Residuals...

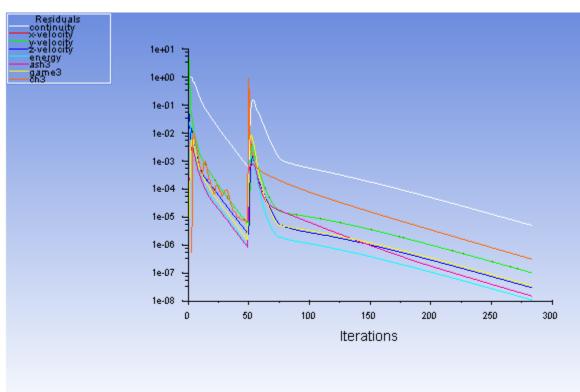
Residual Monitors					×
Options	Equations				
Print to Console	Residual	Monito	r Check Converge	ence Absolute Criteria	<u> </u>
V Plot	continuity	<b>v</b>	<b>V</b>	5e-06	Ξ
Window	x-velocity	<b>V</b>	<b>V</b>	0.001	
1 Curves Axes	y-velocity	<b>V</b>	$\checkmark$	0.001	
Iterations to Plot	z-velocity	<b>V</b>	$\checkmark$	0.001	
1000 🚖	Residual Values	(THE)	(ma)	Convergence Criterion	•
			Iterations	Convergence Criterion absolute	-
Iterations to Store	Normalize		5	absolute	
1000 🚖	Scale			Convergence Conditio	ns
Compute Local Scale					
OK Plot Renormalize Cancel Help					

a. Enter 5e-06 for Absolute Criteria for continuity.

- b. Click **OK** to close the **Residual Monitors** dialog box.
- 7. Request 200 more iterations.

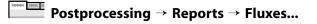
## **Solving** $\rightarrow$ Run Calculation $\rightarrow$ Calculate

The solution will converge in approximately 175 additional iterations.



#### Figure 15.4: Scaled Residuals

8. Check the mass fluxes.



E Flux Reports		<b>—</b>
Options Mass Flow Rate Total Heat Transfer Rate Total Sensible Heat Transfer Rate Radiation Heat Transfer Rate	Boundaries Filter Text	Results -4.658460323362996e-05 5.5962763594196e-05 -9.378143422831282e-06
	4	< ► Net Results (kg/s)
Save Output Parameter		1.693773e-11
	Compute Write Close Help	h.

- a. Retain the default selection of **Mass Flow Rate** in the **Options** group box.
- b. Retain the selection of outlet and velocity-inlet and, wall-4 from the Boundaries selection list.
- c. Click **Compute**, examine the values displayed in the **Results** and **Net Results** boxes, and close the **Flux Reports** dialog box.

Again, the net mass imbalance should be a small fraction (for example, 0.5% or less) of the total flux through the system.

9. Save the case and data files (surface-react1.cas.gz and surface-react1.dat.gz).

**File**  $\rightarrow$  Write  $\rightarrow$  Case & Data...

### **15.4.10.** Postprocessing the Solution Results

1. Create an iso-surface near wall-4.

Postprocessing  $\rightarrow$  Surface  $\rightarrow$  Create  $\rightarrow$  Iso-Surface...

Iso-Surface	<b>—</b>
Surface of Constant Mesh	From Surface Filter Text
Z-Coordinate         ~           Min (m)         Max (m)           0         0.454           Iso-Values (m)	default-interior outlet velocity-inlet wall-1 wall-2 wall-4
0.075438	From Zones Fiter Text
New Surface Name z=0.07	] fluid
Create Compute	Manage Close Help

- a. Select Mesh... and Z-Coordinate from the Surface of Constant drop-down lists.
- b. Click Compute.

The **Min** and **Max** fields display the z-extent of the domain.

- c. Enter 0.075438 m for Iso-Values.
- d. Enter z=0.07 for New Surface Name.

#### Note

If you want to delete or otherwise manipulate any surfaces, click **Manage..** to open the **Surfaces** dialog box.

e. Click Create and close the Iso-Surface dialog box.

The new surface z=0.07 is added to the surfaces selection list.

2. Display contours of temperature on the plane surface created. (Figure 15.5: Temperature Contours Near wall-4 (p. 653)).

Postprocessing  $\rightarrow$  Graphics  $\rightarrow$  Contours  $\rightarrow$  Edit...

Contours			×
Options Filled	Contours of Temperature		•
<ul> <li>Node Values</li> <li>Global Range</li> </ul>	Static Temperature		•
Auto Range	Min (k)	Max (k)	
Clip to Range	293	1023	
Draw Profiles Draw Mesh	Surfaces Filter Text		<b>x</b> -1
Coloring Banded Smooth Levels Setup 20 1 +	velocity-inlet wall-1 wall-2 wall-4 wall-5 wall-6 z=0.07 New Surface ▼ Display Compute	e Close Help	

- a. Ensure that **Filled** is enabled in the **Options** group box.
- b. Select Temperature... and Static Temperature from the Contours of drop-down lists.
- c. Deselect wall-4 from the Surfaces selection list.
- d. Select **z=0.07** from the **Surfaces** selection list.
- e. Click Display.

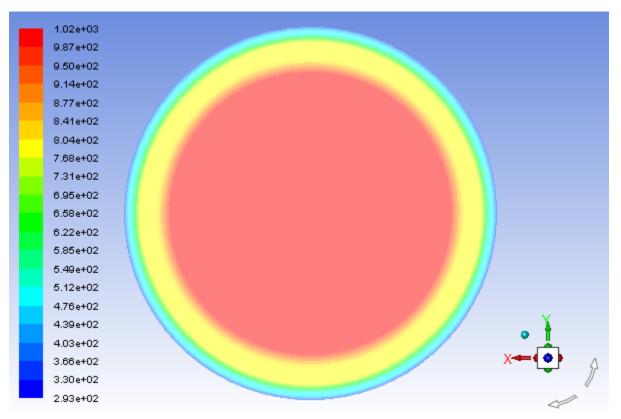


Figure 15.5: Temperature Contours Near wall-4

*Figure 15.5: Temperature Contours Near wall-4(p. 653) shows the temperature distribution across a plane just above the rotating disk. You can see that the disk has a temperature of 1023 K.* 

3. Display contours of surface deposition rates of **ga** (Figure 15.6: Contours of Surface Deposition Rate of ga (p. 654)).

Postprocessing  $\rightarrow$  Graphics  $\rightarrow$  Contours  $\rightarrow$  Edit...

- a. Select Species... and Surface Deposition Rate of ga from the Contours of drop-down lists.
- b. Select wall-4 from the Surfaces selection list.
- c. Deselect **z=0.07** from the **Surfaces** selection list.
- d. Click Display.

You may need to use the left mouse button to rotate the image so that you can see the contours on the top side of **wall-4** where the deposition takes place.

*Figure 15.6: Contours of Surface Deposition Rate of ga (p. 654) shows the gradient of surface deposition rate of ga. The maximum deposition is seen at the center of the disk.* 

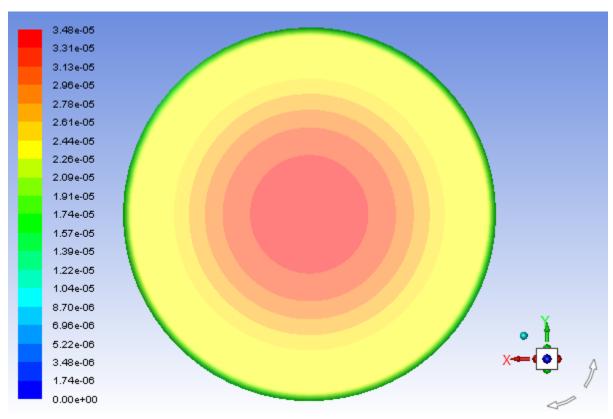


Figure 15.6: Contours of Surface Deposition Rate of ga

4. Display contours of surface coverage of **ga\_s** (Figure 15.7: Contours of Surface Coverage of ga\_s (p. 655)).

Postprocessing  $\rightarrow$  Graphics  $\rightarrow$  Contours  $\rightarrow$  Edit...

- a. Select **Species...** and **Surface Coverage of ga\_s** from the **Contours of** drop-down lists.
- b. Retain the selection of **wall-4** in the **Surfaces** selection list.
- c. Click **Display** and close the **Contours** dialog box.

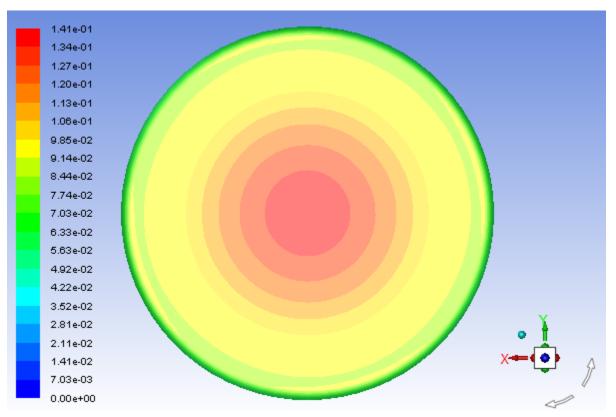


Figure 15.7: Contours of Surface Coverage of ga\_s

Figure 15.7: Contours of Surface Coverage of ga\_s (p. 655) shows the rate of surface coverage of the site species **ga\_s**.

5. Create a line surface from the center of **wall-4** to the edge.

💶 Line/	Rake Surface		×	
Option Cline Res	Tool Type		Number of Points	
-End Po	pints			
x0 (m)	-0.01040954	x1 (m)	0.1428	
y0 (m)	-0.004949478	y1 (m)	0.1386585	
z0 (m)	z0 (m) 0.0762001		0.07620001	
	Select Points with Mouse			
	New Surface Name			
line-9				
Create Manage Close Help				

Postsprocessing  $\rightarrow$  Surface  $\rightarrow$  Create  $\rightarrow$  Line/Rake...

a. Enter the values for **x0**, **x1**, **y0**, **y1**, **z0**, and **z1** as follows:

End Points	Value
x0	-0.01040954
уО	-0.004949478
z0	0.0762001
x1	0.1428
у1	0.1386585
z1	0.07620001

You can also select the points by clicking **Select Points with Mouse**. Then, in the graphic display, click at the center of **wall-4** and at the edge using the right mouse button.

b. Click **Create** to accept the default name of line-9 for the **New Surface Name**.

#### Note

If you want to delete or otherwise manipulate any surfaces, click **Manage...** to open the **Surfaces** dialog box

- c. Close the Line/Rake Surface dialog box.
- 6. Plot the surface deposition rate of Ga versus radial distance (Figure 15.8: Plot of Surface Deposition Rate of Ga (p. 657)).

Postprocessing $\rightarrow$ Plots $\rightarrow$ XY Plot $\rightarrow$ Edit.
--

Solution XY Plot		<b>—</b>	
Solution XY Plot Options Node Values Position on X Axis Position on Y Axis Write to File Order Points File Data	Plot Direction X 1 Y 0 Z 0	Y Axis Function Species Surface Deposition Rate of ga X Axis Function Direction Vector Surfaces Filter Text default-interior line-9 outlet velocity-inlet wall-1 wall-2	
wall-4 			

- a. Disable Node Values in the Options group box.
- b. Select Species... and Surface Deposition Rate of ga from the Y Axis Function drop-down lists.

The source/sink terms due to the surface reaction are deposited in the cell adjacent to the wall cells, so it is necessary to plot the cell values and not the node values.

- c. Select line-9 you just created from the Surfaces selection list.
- d. Click Plot and close the Solution XY Plot dialog box.

The peak surface deposition rate occurs at the center of **wall-4** (where the concentration of the mixture *is highest*).

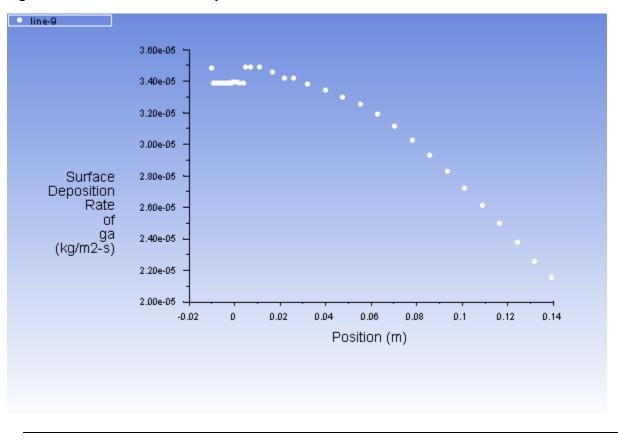


Figure 15.8: Plot of Surface Deposition Rate of Ga

#### Extra

You can also perform all the postprocessing steps to analyze the deposition of As.

7. Save the case and data files (surface-react2.cas.gz and surface-react2.dat.gz).

```
File \rightarrow Write \rightarrow Case & Data...
```

## 15.5. Summary

The main focus of this tutorial is the accurate modeling of macroscopic gas flow, heat and mass transfer, species diffusion, and chemical reactions (including surface reactions) in a rotating disk CVD reactor. In

this tutorial, you learned how to use the two-step surface reactions involving site species, and computed simultaneous deposition of gallium and arsenide from a mixture of precursor gases on a rotating susceptor. Note that the same approach is valid if you are simulating multi-step reactions with multiple sites/site species.

## **15.6. Further Improvements**

This tutorial guides you through the steps to reach an initial solution. You may be able to obtain a more accurate solution by using an appropriate higher-order discretization scheme and by adapting the mesh. Mesh adaption can also ensure that the solution is independent of the mesh. These steps are demonstrated in Introduction to Using ANSYS Fluent: Fluid Flow and Heat Transfer in a Mixing Elbow (p. 121).

# **Chapter 16: Modeling Evaporating Liquid Spray**

This tutorial is divided into the following sections:

16.1. Introduction
16.2. Prerequisites
16.3. Problem Description
16.4. Setup and Solution
16.5. Summary
16.6. Further Improvements

# 16.1. Introduction

In this tutorial, the air-blast atomizer model in ANSYS Fluent is used to predict the behavior of an evaporating methanol spray. Initially, the air flow is modeled without droplets. To predict the behavior of the spray, the discrete phase model is used, including a secondary model for breakup.

This tutorial demonstrates how to do the following:

- Define a spray injection for an air-blast atomizer.
- Calculate a solution using the discrete phase model in ANSYS Fluent.

# 16.2. Prerequisites

This tutorial is written with the assumption that you have completed one or more of the introductory tutorials found in this manual:

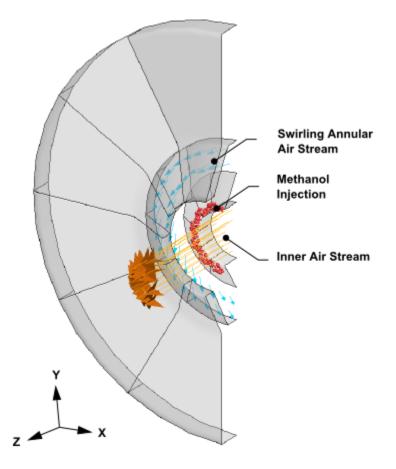
- Introduction to Using ANSYS Fluent in ANSYS Workbench: Fluid Flow and Heat Transfer in a Mixing Elbow (p. 1)
- Parametric Analysis in ANSYS Workbench Using ANSYS Fluent (p. 73)
- Introduction to Using ANSYS Fluent: Fluid Flow and Heat Transfer in a Mixing Elbow (p. 121)

and that you are familiar with the ANSYS Fluent tree and ribbon structure. Some steps in the setup and solution procedure will not be shown explicitly.

## **16.3. Problem Description**

The geometry to be considered in this tutorial is shown in Figure 16.1: Problem Specification (p. 660). Methanol is cooled to  $-10^{\circ}$ C before being introduced into an air-blast atomizer. The atomizer contains an inner air stream surrounded by a swirling annular stream. To make use of the periodicity of the problem, only a 30° section of the atomizer will be modeled.

#### Figure 16.1: Problem Specification



## 16.4. Setup and Solution

The following sections describe the setup and solution steps for this tutorial:

16.4.1. Preparation
16.4.2. Mesh
16.4.3. Solver
16.4.4. Models
16.4.5. Materials
16.4.6. Boundary Conditions
16.4.7. Initial Solution Without Droplets
16.4.8. Creating a Spray Injection
16.4.9. Solution
16.4.10. Postprocessing

## 16.4.1. Preparation

To prepare for running this tutorial:

1. Set up a working folder on the computer you will be using.

2. Go to the ANSYS Customer Portal, https://support.ansys.com/training.

#### Note

If you do not have a login, you can request one by clicking **Customer Registration** on the log in page.

- 3. Enter the name of this tutorial into the search bar.
- 4. Narrow the results by using the filter on the left side of the page.
  - a. Click ANSYS Fluent under Product.
  - b. Click **18.0** under **Version**.
- 5. Select this tutorial from the list.
- 6. Click the evaporate\_liquid\_R180.zip link to download the input files.
- 7. Unzip evaporate\_liquid\_R180.zip to your working folder.

The mesh file sector.msh can be found in the evaporate\_liquid directory created after unzipping the file.

8. Use the ANSYS Fluent Launcher to start the **3D** version of ANSYS Fluent.

Fluent Launcher displays your **Display Options** preferences from the previous session.

For more information about the ANSYS Fluent Launcher, see starting ANSYS Fluent using the Fluent Launcher in the Fluent Getting Started Guide.

- 9. Ensure that the **Display Mesh After Reading** and **Workbench Color Scheme** options are enabled.
- 10. Ensure that the Serial processing option is selected.
- 11. Enable **Double Precision**.
- 12. Ensure that **Meshing Mode** is disabled.

### 16.4.2. Mesh

1. Read in the mesh file sector.msh.

**File**  $\rightarrow$  Read  $\rightarrow$  Mesh...

2. Change the periodic type of **periodic-a** to rotational.

**E** Setup  $\rightarrow$  **\bigcirc** Boundary Conditions  $\rightarrow \stackrel{=}{\equiv}$  periodic-a  $\rightarrow$  Edit...

Periodic		×
Zone Name periodic-a		
Periodic Type Translational Rotational		
	OK Cancel Help	

- a. Select Rotational in the Periodic Type group box.
- b. Click **OK** to close the **Periodic** dialog box.
- 3. In a similar manner, change the periodic type of **periodic-b** to rotational.
- 4. Check the mesh.



**Setting Up Domain**  $\rightarrow$  Mesh  $\rightarrow$  Display...

ANSYS Fluent will perform various checks on the mesh and report the progress in the console. Ensure that the reported minimum volume is a positive number.

5. Display the mesh.

💶 Mesh Display		
Options E Nodes Edges Faces Partitions Overset Shrink Factor Feat 0 20	All     Feature     Outline ture Angle terior	Surfaces Filter Text
Display Colors Close Help		

a. Ensure that **Faces** is enabled in the **Options** group box.

b. Select only atomizer-wall, central\_air, and swirling\_air from the Surfaces selection list.

Tip

To deselect all surfaces click the far-right button  $\square$  at the top of the **Surfaces** selection list, and then select the desired surfaces from the **Surfaces** selection list.

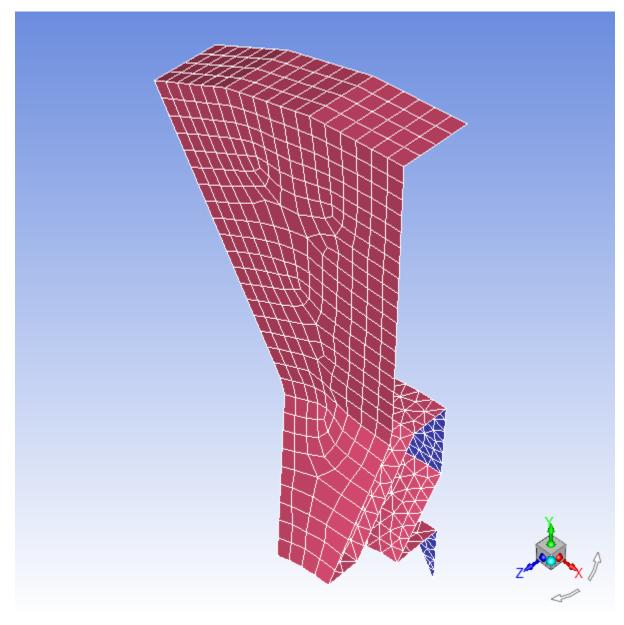
c. Click the **Colors...** button to open the **Mesh Colors** dialog box.

Mesh Colors		×		
Options Oclor by Type	Types far-field inlet	Colors		
Color by ID	interior outlet periodic rans-les-interface	light red light yellow magenta ■ maroon		
	symmetry axis wall	orange pink red		
	free-surface internal traction	vhite vellow v		
Reset Colors Close Help				

- i. Select wall from the Types selection list.
- ii. Select **pink** from the **Colors** selection list.
- iii. Close the **Mesh Colors** dialog box.
- d. Click **Display** and close the **Mesh Display** dialog box.

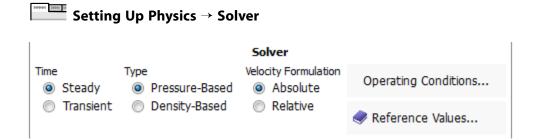
The graphics display will be updated to show the mesh. Zoom in with the mouse to obtain the view shown in Figure 16.2: Air-Blast Atomizer Mesh Display (p. 664).

Figure 16.2: Air-Blast Atomizer Mesh Display



### 16.4.3. Solver

Retain the default solver settings of pressure-based steady-state solver in the **Solver** group of the **Setting Up Physics** tab.



## 16.4.4. Models

1. Enable heat transfer by enabling the energy equation.

Setting Up Physics $\rightarrow$ Models $\rightarrow$ Energy					
Models					
	Radiation	💽 Multiphase	🙆 Solidify/Melt		
Energy	∦ <sub>≠</sub> Heat Exchanger	Species	动) Acoustics		
	Viscous	📑 Discrete Phase	🗄 More 🗸		

2. Enable the Realizable k- $\varepsilon$  turbulence model.

<b>E</b> Viscous Model			
Model	Model Constants		
Inviscid	C2-Epsilon		
🔘 Laminar	1.9		
Spalart-Allmaras (1 eqn)	TKE Prandtl Number		
k-epsilon (2 eqn)	1		
<ul> <li>k-omega (2 eqn)</li> <li>Transition k-kl-omega (3 eqn)</li> </ul>	TDR Prandtl Number		
Transition SST (4 eqn)	1.2		
<ul> <li>Reynolds Stress (7 eqn)</li> </ul>	Energy Prandtl Number		
<ul> <li>Scale-Adaptive Simulation (SAS)</li> </ul>	0.85		
<ul> <li>Detached Eddy Simulation (DES)</li> <li>Large Eddy Simulation (LES)</li> </ul>	Wall Prandtl Number		
	0.85		
k-epsilon Model			
Standard			
<ul> <li>RNG</li> <li>Realizable</li> </ul>	User-Defined Functions		
	Turbulent Viscosity		
Near-Wall Treatment	none 🔻		
Standard Wall Functions	Prandtl Numbers		
Scalable Wall Functions	TKE Prandtl Number		
<ul> <li>Non-Equilibrium Wall Functions</li> <li>Enhanced Wall Treatment</li> </ul>	none		
<ul> <li>Ennanced Wall Treatment</li> <li>Menter-Lechner</li> <li>User-Defined Wall Functions</li> </ul>	TDR Prandtl Number		
	none 🔻		
- ···	Energy Prandtl Number		
Options	none		
<ul> <li>Viscous Heating</li> <li>Curvature Correction</li> <li>Production Limiter</li> </ul>	Wall Prandtl Number		
	none 🔻		
OK Cancel Help			

- a. Select **k-epsilon (2 eqn)** in the **Model** list.
- b. Select Realizable in the k-epsilon Model list.

The Realizable  $k - \varepsilon$  model gives a more accurate prediction of the spreading rate of both planar and round jets than the standard  $k - \varepsilon$  model.

- c. Retain the default selection of Standard Wall Functions in the Near-Wall Treatment list.
- d. Click **OK** to close the **Viscous Model** dialog box.
- 3. Enable chemical species transport and reaction.

```
Setting Up Physics \rightarrow Models \rightarrow Species...
```

Species Model					
Model Off Species Transport Non-Premixed Combustion Premixed Combustion Partially Premixed Combustion Composition PDF Transport	Mixture Properties Mixture Material methyl-alcohol-air Import CHEMKIN Mechanism Number of Volumetric Species 5				
Reactions Volumetric					
Options Inlet Diffusion Diffusion Energy Source Full Multicomponent Diffusion Thermal Diffusion	Select Boundary Species Select Monitored Species				
OK Apply Cancel Help					

a. Select Species Transport in the Model list.

#### b. Select methyl-alcohol-air from the Mixture Material drop-down list.

The **Mixture Material** list contains the set of chemical mixtures that exist in the ANSYS Fluent database. When selecting an appropriate mixture for your case, you can review the constituent species and the reactions of the predefined mixture by clicking **View...** next to the **Mixture Material** drop-down list. The chemical species and their physical and thermodynamic properties are defined by the selection of the mixture material. After enabling the **Species Transport** model, you can alter the mixture material selection or modify the mixture material properties using the **Create/Edit Materials** dialog box. You will modify your local copy of the mixture material later in this tutorial.

c. Click **OK** to close the **Species Model** dialog box.

## 16.4.5. Materials

Define materials using the **Materials** task page.

E Setup → ♀Materials

Materials
Materials
Mixture methyl-alcohol-air nitrogen water-vapor carbon-dioxide oxygen methyl-alcohol-vapor Fluid air Solid aluminum
Create/Edit Delete
Help

1. Remove water vapor and carbon dioxide from the **Mixture Species** list.

**E** Setup  $\rightarrow$   $\clubsuit$  Materials  $\rightarrow \equiv$  Mixture  $\rightarrow$  Create/Edit...

ame	Material Type	Order Materials by
methyl-alcohol-air	mixture	Name     Chemical Formula
hemical Formula	Fluent Mixture Materials	Chemical Pormula
	methyl-alcohol-air	<ul> <li>Fluent Database</li> </ul>
	Mixture	User-Defined Database
	none	T
roperties		
Mixture Species	names	
Density (kg/m3)	incompressible-ideal-gas	
Cp (Specific Heat) (j/kg-k)	mixing-law	
Thermal Conductivity (w/m-k)	constant	
	0.0454	

a. Click the **Edit** button next to the **Mixture Species** drop-down list to open the **Species** dialog box.

Species	×
Mixture methyl-alcohol-air	
Available Materials	Selected Species
air carbon-dioxide (co2) water-vapor (h2o)	ch3oh o2 n2
Selected Site Species	Add Remove Selected Solid Species
Add Remove	Add Remove
ОК Са	Help

- i. Select carbon dioxide (co2) from the Selected Species selection list.
- ii. Click **Remove** to remove carbon dioxide from the **Selected Species** list.

- iii. In a similar manner, remove water vapor (h2o) from the Selected Species list.
- iv. Click OK to close the Species dialog box.
- b. Click Change/Create and close the Create/Edit Materials dialog box.

Note

It is good practice to click the **Change/Create** button whenever changes are made to material properties even though it is not necessary in this case.

## 16.4.6. Boundary Conditions

Specify boundary conditions using the **Boundary Conditions** task page.

Boundary Condit	tions	
Zone Filter Text		] 루
atomizer-wall		
central_air		
co-flow-air		
default-interior		
outer-wall		
outlet		
periodic-a periodic-b		
swirling_air		
swining_an		
Phase Type	ID	
	s-flow-inlet	
Edit	Copy Profiles	
Parameters	Operating Conditions	
Display Mesh		
	Periodic Conditions	
🔲 Highlight Zone		
Help		
(Help)		

1. Set the boundary conditions for the inner air stream (central\_air).

Mass-Flow Inlet		×				
Zone Name						
central_air						
Momentum Thermal Radiation	Species DPM Multip	hase Potential UDS				
Reference Frame Abs	olute	•				
Mass Flow Specification Method Mas	s Flow Rate	•				
Mass Flow Rate (kg/s)	9.167e-5	constant 🔹				
Supersonic/Initial Gauge Pressure (pascal)	0	constant 🔹				
Direction Specification Method Dire	ction Vector	•				
Coordinate System Cart	esian (X, Y, Z)	•				
X-Component of Flow Direction	0	constant 🔹				
Y-Component of Flow Direction	0	constant 🔹				
Z-Component of Flow Direction	1	constant 🔹				
Turbulence						
Specification Method Inter	nsity and Hydraulic Diameter	· •				
	Turbulent Intensity (%) 10	P				
	Hydraulic Diameter (m) 0.0	0037 P				
OK Cancel Help						

# **E** Setup $\rightarrow$ **\bigcirc** Boundary Conditions $\rightarrow$ **\stackrel{\frown}{=}** central\_air $\rightarrow$ Edit...

- a. Enter 9.167e-5 kg/s for Mass Flow Rate.
- b. Enter 0 for X-Component of Flow Direction.
- c. Retain the default value of 0 for **Y-Component of Flow Direction**.
- d. Enter 1 for **Z-Component of Flow Direction**.
- e. Select Intensity and Hydraulic Diameter from the Specification Method drop-down list.
- f. Enter 10 for **Turbulent Intensity**.
- g. Enter 0.0037 m for Hydraulic Diameter.
- h. Click the Thermal tab and enter 293 K for Total Temperature.
- i. Click the Species tab and enter 0.23 for o2 in the Species Mass Fractions group box.
- j. Click **OK** to close the **Mass-Flow Inlet** dialog box.
- 2. Set the boundary conditions for the air stream surrounding the atomizer (co-flow-air).

# **Setup** $\rightarrow$ **Conditions** $\rightarrow \stackrel{\bullet}{=}$ **Co-flow-air** $\rightarrow$ **Edit...**

Velocity Inlet								×
Zone Name								
co-flow-air								
Momentum	Thermal	Radiation	Species	DPM	Multiph	nase	Potential	UDS
Velocity	Specification	Method Mag	initude, Nor	mal to Bo	oundary			•
	Reference	e Frame Abs	olute					•
	Velocity Mag	nitude (m/s)	1			const	ant	•
Supersonic/Initi	ial Gauge Pres	ssure (pascal)	0			const	ant	•
	Turbulence							
5	Specification I	Method Inter	nsity and Hy	draulic D	iameter			•
			Turbulent I	ntensity	(%) 5			P
			Hydraulic D	)iameter	(m) 0.0	726		P
OK Cancel Help								

- a. Enter 1 m/s for **Velocity Magnitude**.
- b. Select Intensity and Hydraulic Diameter from the Specification Method drop-down list.
- c. Retain the default value of 5 for **Turbulent Intensity**.
- d. Enter 0.0726 m for Hydraulic Diameter.
- e. Click the **Thermal** tab and enter 293 K for **Temperature**.
- f. Click the Species tab and enter 0.23 for o2 in the Species Mass Fractions group box.
- g. Click OK to close the Velocity Inlet dialog box.
- 3. Set the boundary conditions for the exit boundary (**outlet**).

**E** Setup  $\rightarrow$  **\bigcirc** Boundary Conditions  $\rightarrow$  **\stackrel{\frown}{=}** outlet  $\rightarrow$  Edit...

💶 Pre	essure Out	let						<b>×</b>
Zone	Name							
outlet	t							
Mor	mentum	Thermal	Radiation	Species	DPM	Multiphase	e Potential	UDS
	В	ackflow Refe	rence Frame	Absolute				•
		Gauge F	Pressure (pas	cal) 0			constant	•
Backf	flow Direc	tion Specifica	tion Method	From Neigh	boring Ce	ell		•
	Backfl	ow Pressure	Specification	Total Pressu	ire			•
🗖 Ra	adial Equili	brium Pressu	re Distributior	n				
🗖 A	verage Pr	essure Specif	ication					
Ta	arget Mass	s Flow Rate						
		- Turbuler	ice					
		Specificat	ion Method	Intensity and	l Viscosit	y Ratio		<b></b>
			Backfl	ow Turbulen	t Intensi	ty (%) 5		P
			Backflo	w Turbulent	Viscosit	y Ratio 5		P
	OK Cancel Help							

- a. Select From Neighboring Cell from the Backflow Direction Specification Method drop-down list.
- b. Retain Intensity and Viscosity Ratio from the Specification Method drop-down list.
- c. Retain the default value of 5 for **Backflow Turbulent Intensity (%)**.
- d. Enter 5 for Backflow Turbulent Viscosity Ratio.
- e. Click the Thermal tab and enter 293 K for Backflow Total Temperature.
- f. Click the Species tab and enter 0.23 for o2 in the Species Mass Fractions group box.
- g. Click **OK** to close the **Pressure Outlet** dialog box.
- 4. Set the boundary conditions for the swirling annular stream (swirling\_air).

**Setup**  $\rightarrow$  **Conditions**  $\rightarrow$  **Setup**  $\rightarrow$  **Setup**  $\rightarrow$  **Edit...** 

💶 Velocity Inle	et .		<b>X</b>			
Zone Name						
swirling_air						
Momentum	Thermal Radiation	Species DPM Multipl	nase Potential UDS			
Velocit	ty Specification Method Mag	nitude and Direction	•			
	Reference Frame Abso	olute	•			
	Velocity Magnitude (m/s)	19	constant 🔹			
Supersonic/In	itial Gauge Pressure (pascal)	0	constant 🔹			
	Coordinate System Cylin	drical (Radial, Tangential, Ax	ial) 🔻			
Radial-C	omponent of Flow Direction	0	constant 🔹			
Tangential-C	omponent of Flow Direction	0.7071	constant 🔻			
Axial-C	omponent of Flow Direction	0.7071	constant 🔹			
	Turbulence					
	Specification Method Inter	isity and Hydraulic Diameter	•			
		Turbulent Intensity (%) 5	P			
		Hydraulic Diameter (m) 0.0	043 P			
OK Cancel Help						

- a. Select Magnitude and Direction from the Velocity Specification Method drop-down list.
- b. Enter 19 m/s for Velocity Magnitude.
- c. Select Cylindrical (Radial, Tangential, Axial) from the Coordinate System drop-down list.
- d. Enter 0 for Radial-Component of Flow Direction.
- e. Enter 0.7071 for Tangential-Component of Flow Direction.
- f. Enter 0.7071 for Axial-Component of Flow Direction.
- g. Select Intensity and Hydraulic Diameter from the Specification Method drop-down list.
- h. Retain the default value of 5 for Turbulent Intensity.
- i. Enter 0.0043 m for Hydraulic Diameter.
- j. Click the **Thermal** tab and enter 293 K for **Temperature**.
- k. Click the Species tab and enter 0.23 for o2 in the Species Mass Fractions group box.
- I. Click **OK** to close the **Velocity Inlet** dialog box.
- 5. Set the boundary conditions for the outer wall of the atomizer (**outer-wall**).

Setup →		Conditions $\rightarrow \overline{E}$	outer-wall →	Edit
---------	--	---------------------------------------	--------------	------

💶 Wall							×
Zone Name							
outer-wall							
Adjacent Cell Zone fluid							
Momentum Thermal	Radiation	Species	DPM	Multiphase	UDS	Wall Film	Potential
Wall Motion     Motion     Stationary Wall     Relative to Adjacent Cell Zone     Moving Wall							
Shear Condition No Slip Specified Shear	Shear Stre X-Compone	ess ent (pascal)	0		CO	nstant	•
<ul> <li>Specified Shear</li> <li>Specularity Coefficient</li> <li>Marangoni Stress</li> </ul>	Y-Component (pascal) 0 Z-Component (pascal) 0				constant   constant		
Wall Roughness Roughness Models	-Sand-Gr	ain Roughne	955				
<ul> <li>Standard</li> <li>High Roughness (Icing)</li> </ul>		ess Height (				constant	•
	Rougr	ness Consta	ant 0.5			constant	•
OK Cancel Help							

- a. Select Specified Shear in the Shear Condition list.
- b. Retain the default values for the remaining parameters.
- c. Click **OK** to close the **Wall** dialog box.

## 16.4.7. Initial Solution Without Droplets

The airflow will first be solved and analyzed without droplets.

1. Set the solution method.



Solution Methods	
Pressure-Velocity Coupling	
Scheme	
Coupled	
Spatial Discretization	
Gradient	-
Least Squares Cell Based	
Pressure	
Second Order	Ξ
Momentum	
Second Order Upwind 🔹	
Turbulent Kinetic Energy	
First Order Upwind 🔹	
Turbulent Dissipation Rate	
First Order Upwind	-
Transient Formulation	
Non-Iterative Time Advancement	
Frozen Flux Formulation	
Pseudo Transient	
Warped-Face Gradient Correction	
High Order Term Relaxation Options	
Set All Species Discretizations Together	
Default	
Help	

- a. Select **Coupled** from the **Scheme** drop-down list in the **Pressure-Velocity Coupling** group box.
- b. Enable **Pseudo Transient**.

The message appears in the console informing you of changing AMG cycle type for Volume Fraction, Turbulent Kinetic Energy, and Turbulent Dissipation Rate to F-cycle.

2. Retain the default under-relaxation factors.



#### Solution Controls

Pressure	
0.5	
Momentum	
0.5	_ 1
0.5	
Density	
1	
Body Forces	_
1	
Furbulent Kinetic Energy	
0.75	_
efault	
quations] Limits] Advanced]	
Set All Species URFs Together	

3. Enable residual plotting during the calculation.

Solving $\rightarrow$ Reports $\rightarrow$ Residuals
---

Residual Monitors					<b>x</b>
Options  Print to Console  Plot  Window  1  Curves  Iterations to Plot  1000	Equations Residual continuity x-velocity y-velocity z-velocity	Monitor C	Check Convergence	Absolute Criteria 0.001 0.001 0.001 0.001 0.001	
Iterations to Store	Residual Values           Normalize           Scale           Compute Local	al Scale	Iterations	Convergence Co absolute	riterion
OK Plot Renormalize Cancel Help					

a. Ensure that **Plot** is enabled in the **Options** group box.

- b. Click **OK** to close the **Residual Monitors** dialog box.
- 4. Initialize the flow field.

Solving → Initialization					
	Initializ	ation			
Method		Patch			
O Hybrid	More Settings	Reset Statistics			
Standard	Options	Decet DDM	t = 0 Initialize		
		Reset DPM	Inicialize		

- a. Retain the **Method** at the default of **Hybrid**.
- b. Click Initialize to initialize the variables.

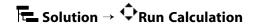
#### Note

For flows in complex topologies, hybrid initialization will provide better initial velocity and pressure fields than standard initialization. This will help to improve the convergence behavior of the solver.

5. Save the case file (spray1.cas.gz).



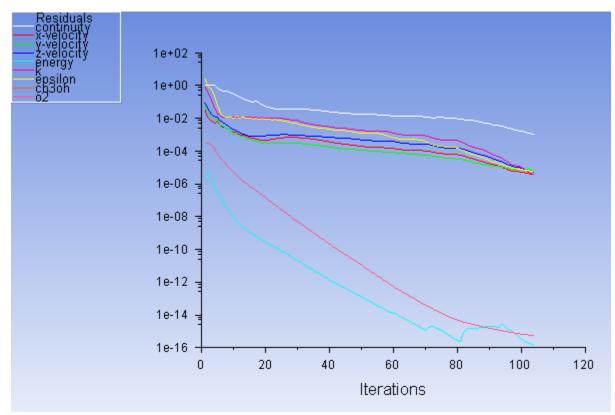
6. Start the calculation by requesting 150 iterations.



- a. Select User Specified from the Time Step Method group box.
- b. Retain 1 s for **Pseudo Time Step**.
- c. Enter 150 for Number of Iterations.
- d. Click Calculate.

The solution will converge in approximately 105 iterations.

#### Figure 16.3: Scaled Residuals



7. Save the case and data files (spray1.cas.gz and spray1.dat.gz).

File  $\rightarrow$  Write  $\rightarrow$  Case & Data... н

#### Note

ANSYS Fluent will ask you to confirm that the previous case file is to be overwritten.

8. Create a clip plane to examine the flow field at the midpoint of the atomizer section.



So-Surface	
Surface of Constant Mesh	From Surface Filter Text
Angular Coordinate 👻	atomizer-wall
Min (deg) Max (deg) 0 30 Iso-Values (deg) 15	central_air co-flow-air default-interior outer-wall outlet From Zones Filter Text
New Surface Name angle=15	fluid
Create Compute	Manage Close Help

- a. Select Mesh... and Angular Coordinate from the Surface of Constant drop-down lists.
- b. Click **Compute** to obtain the minimum and maximum values of the angular coordinate.
- c. Enter 15 for Iso-Values.
- d. Enter angle=15 for New Surface Name.
- e. Click **Create** to create the isosurface.
- f. Close the **Iso-Surface** dialog box.
- 9. Review the current state of the solution by examining contours of velocity magnitude (Figure 16.4: Velocity Magnitude at Mid-Point of Atomizer Section (p. 682)).

Postprocessing  $\rightarrow$  Graphics  $\rightarrow$  Contours  $\rightarrow$  Edit...

Contours	
Options Filled	Contours of Velocity
Vode Values	Velocity Magnitude
<ul> <li>Global Range</li> <li>Auto Range</li> </ul>	Min (m/s) Max (m/s)
Clip to Range	0 97.18853
Draw Profiles Draw Mesh	Surfaces Filter Text
Coloring Banded Smooth Levels Setup 20 1	angle=15       ▲         atomizer-wall       ■         central_air       ■         co-flow-air       ■         default-interior       ●         outlet       ▼         New Surface       ▼         Display       Compute       Close         Help

- a. Enable Filled in the Options group box
- b. Select Velocity... and Velocity Magnitude from the Contours of drop-down lists.
- c. Enable Draw Mesh.

The **Mesh Display** dialog box will open.

- i. Retain the current mesh display settings.
- ii. Close the Mesh Display dialog box.
- d. Select angle=15 from the Surfaces selection list.
- e. Click **Display** and close the **Contours** dialog box.
- f. Use the mouse to obtain the view shown in Figure 16.4: Velocity Magnitude at Mid-Point of Atomizer Section (p. 682).

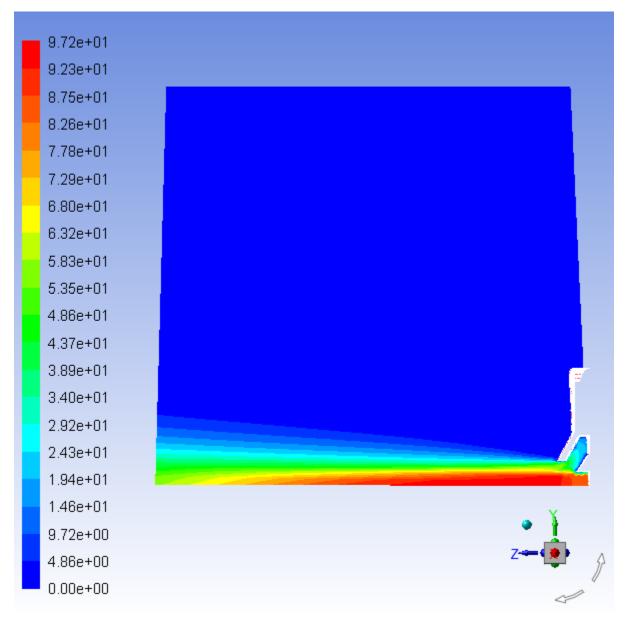


Figure 16.4: Velocity Magnitude at Mid-Point of Atomizer Section

10. Modify the view to include the entire atomizer.

 $\blacksquare \forall Viewing \rightarrow Display \rightarrow Views...$ 

Views	Actions	Mirror Planes [0/0] 🗐 🛒 🗮
back bottom front isometric left right top Save Name view-0	Default Auto Scale Previous Save Delete Read Write	Define Plane Periodic Repeats Define
Ар	Camera	Close Help

a. Click the **Define...** button to open the **Graphics Periodicity** dialog box.

Craphics Period	icity	×
Cell Zones Filter T	ext 🗖	Associated Surfaces Filter Text
fluid		angle=15
		atomizer-wall
		central_air co-flow-air
		default-interior
		fluid 👻
Periodic Type	Axis Direction	Axis Origin
Translational	X (m) 0	X (m) 0
Rotational	Y (m) 0	Y (m) 0
Angle (deg)	Z (m) 1	Z (m) 0
30		
Number of Repeat	s 12 🗘	
	_	
	Se	

- i. Select **fluid** from the **Cell Zones** selection list.
- ii. Retain the selection of Rotational in the Periodic Type list.
- iii. Retain the value of 12 for Number of Repeats.
- iv. Click Set and close the Graphics Periodicity dialog box.

The graphics display will be updated to show the entire atomizer.

b. Click **Apply** and close the **Views** dialog box.

11. Display pathlines of the air in the swirling annular stream (Figure 16.5: Pathlines of Air in the Swirling Annular Stream (p. 685)).

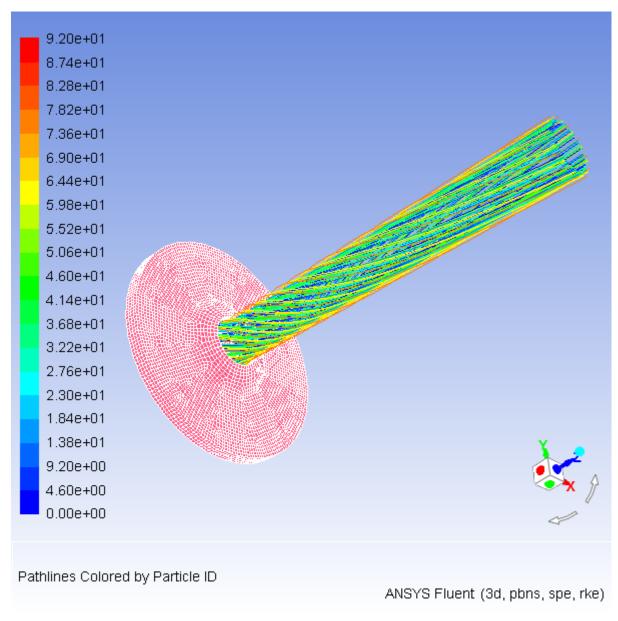
Options Oil Flow	Style		•	Color by Particle Variables		•
Reverse Node Values	Attributes			Particle ID		•
Auto Range	Step Size (m)	Tolerance		Min	Max	
Draw Mesh	0.01	0.001		0	92	
Accuracy Control	Steps	Path Skip				
Relative Pathlines	500 💠	5	<b>\$</b> -	Release from Surfaces	5	× = = = =
XY Plot Write to File	Path Coarsen			swirling_air		
Туре	On Zone					
CFD-Post   Pulse Mode  Continuous  Single	atomizer-wall central_air co-flow-air outer-wall outlet periodic-a periodic-b		*	Highlight Surfaces		

Postprocessing  $\rightarrow$  Graphics  $\rightarrow$  Pathlines  $\rightarrow$  Edit...

- a. Increase the **Path Skip** value to 5.
- b. In the **Release from Surfaces** filter, type **s** to display the surface names that begin with **s** and select **swirling\_air** from the selection list.
- c. Enable **Draw Mesh** in the **Options** group box.

The **Mesh Display** dialog box will open.

- i. Retain the current mesh display settings.
- ii. Close the Mesh Display dialog box.
- d. Click **Display** and close the **Pathlines** dialog box.
- e. Use the mouse to obtain the view shown in Figure 16.5: Pathlines of Air in the Swirling Annular Stream (p. 685).



#### Figure 16.5: Pathlines of Air in the Swirling Annular Stream

16.4.8. Creating a Spray Injection

1. Define the discrete phase modeling parameters.

Setting Up Physics → Models → Discrete Phase...

Discrete Phase Model	<b>—</b>
Interaction	Particle Treatment
Interaction with Continuous Phase	Unsteady Particle Tracking
Update DPM Sources Every Flow Iteration	Track with Fluid Flow Time Step
DPM Iteration Interval 10	Inject Particles at
Contour Plots for DPM Variables	<ul> <li>Particle Time Step</li> <li>Fluid Flow Time Step</li> </ul>
RMS Values	Particle Time Step Size (s) 0.0001
	Number of Time Steps 10
	Clear Particles
Tracking Physical Models UDF Nume	erics Parallel
Tracking Parameters Max. Number of Steps 500	
<ul> <li>Specify Length Scale</li> <li>Step Length Factor</li> <li>5</li> </ul>	
OK Injections DEM C	ollisions Cancel Help

a. Select Interaction with Continuous Phase in the Interaction group box.

This will include the effects of the discrete phase trajectories on the continuous phase.

- b. Retain the value of 10 for **DPM Iteration Interval**.
- c. Select Mean Values in the Contour Plots for DPM Variables group box.

This will make the cell-averaged variables available for postprocessing activities.

- d. Select the Unsteady Particle Tracking option in the Particle Treatment group box.
- e. Enter 0.0001 for Particle Time Step Size.
- f. Enter 10 for **Number of Time Steps**.
- g. Under the **Physical Models** tab, select the **Temperature Dependent Latent Heat** and **Breakup** (**Options** group).

Discrete Phase Model	
Interaction	Particle Treatment
<ul> <li>Interaction with Continuous Phase</li> <li>Update DPM Sources Every Flow Iteration</li> </ul>	<ul> <li>Unsteady Particle Tracking</li> <li>Track with Fluid Flow Time Step</li> </ul>
DPM Iteration Interval 10	Inject Particles at
Contour Plots for DPM Variables	<ul> <li>Particle Time Step</li> <li>Fluid Flow Time Step</li> </ul>
RMS Values	Particle Time Step Size (s) 0.0001 Number of Time Steps 10
	Clear Particles
Tracking Physical Models UDF	Numerics Parallel
Options Thermophoretic Force Saffman Lift Force Virtual Mass Force Pressure Gradient Force Erosion/Accretion Pressure Dependent Boiling Vermperature Dependent Latent Heat Two-Way Turbulence Coupling DEM Collision Stochastic Collision Vermoversity Breakup	Breakup Model Consider Child Particles in the Same Tracking Step
OK Injections D	EM Collisions Cancel Help

h. Under the Numerics tab, select Linearize Source Terms (Source Terms group).

Enabling this option will allow you to run the simulation with more aggressive setting for the **Discrete Phase Sources** under-relaxation factor to speed up the solution convergence.

i. Click Injections... to open the Injections dialog box.

In this step, you will define the characteristics of the atomizer.

💶 Injectio	ns		<b>X</b>
Injections	Filter Text	x=	Create
			Сору
			Delete
			List
			Read
			Write
		Set Close Help	

An **Information** dialog box appears indicating that the **Max. Number of Steps** has been changed from 50000 to 500. Click **OK** in the **Information** dialog box to continue.

j. Click the **Create** button to create the spray injection.

	Set Injection Properties							<b>—</b>
I	njection Name		Inject	ion Type				Number of Streams
Ī	njection-0		air-bla	st-atomizer				• 600 💠
	Particle Type Massless Inert Drop	let 🔿 Combusting 🔿 Multic	ompone	Laws nt 🔲 Custom				
N	laterial	Diameter Distribution	Oxid	izing Species		Discrete Phase D	omain	
	methyl-alcohol-liquid 🔹	linear	¥]		Ψ	none		•
E	vaporating Species	Devolatilizing Species	Proc	luct Species				
	ch3oh 🔹				Ŧ	]		
ſ	Point Properties Physical I	Models Turbulent Dispers	ion P	arcel Wet Com	bustion	Components	UDF	Multiple Reactions
	Start Time (s) Stop Time (s) Injector Inner Diameter (m) Injector Outer Diameter (m) Spray Half Angle (deg) Relative Velocity (m/s)	0 P 100 P 0.0035 P 0.0045 P 45 82.6 P	constan	t			·	Stagger Options Stagger Positions
	Azimuthal Start Angle (deg) Azimuthal Stop Angle (deg)	0 P 30 P					+	
		01	File	. Cancel Help	]			h.

- k. In the **Set Injection Properties** dialog box, select **air-blast-atomizer** from the **Injection Type** dropdown list.
- I. Enter 600 for Number of Streams.

This option controls the number of droplet parcels that are introduced into the domain at every time step.

- m. Select **Droplet** in the **Particle Type** group box.
- n. Select methyl-alcohol-liquid from the Material drop-down list.
- o. In the **Point Properties** tab, specify point properties for particle injections.
  - i. Retain the default values of 0 and 0 for **X-Position** and **Y-Position**.
  - ii. Enter 0.0015 for **Z-Position**.
  - iii. Retain the default values of 0, 0, and 1 for X-Axis, Y-Axis, and Z-Axis, respectively.
  - iv. Enter 263 K for Temperature.

Scroll down the list to see the remaining point properties.

v. Enter 8.5e-5 kg/s for Flow Rate.

This is the methanol flow rate for a 30-degree section of the atomizer. The actual atomizer flow rate is 12 times this value.

vi. Retain the default **Start Time** of 0 s and enter 100 s for the **Stop Time**.

For this problem, the injection should begin at t=0 and not stop until long after the time period of interest. A large value for the stop time (for example, 100 s) will ensure that the injection will essentially never stop.

#### vii. Enter 0.0035 m for the Injector Inner Diameter and 0.0045 m for the Injector Outer Diameter.

viii.Enter 45 degrees for Spray Half Angle.

The spray angle is the angle between the liquid sheet trajectory and the injector centerline.

ix. Enter 82.6 m/s for the **Relative Velocity**.

The relative velocity is the expected relative velocity between the atomizing air and the liquid sheet.

x. Retain the default Azimuthal Start Angle of 0 degrees and enter 30 degrees for the Azimuthal Stop Angle.

This will restrict the injection to the 30-degree section of the atomizer that is being modeled.

p. In the **Physical Models** tab, specify the breakup model and drag parameters.

Injection Name		Injection T	iype			Number of Stre	earm
injection-0		air-blast-at	tomizer			• 600	÷
Particle Type Massless O Inert O Drop	let 🔿 Combusting 🛇 Multico	mponent	Laws				
Material	Diameter Distribution	Oxidizing	Species	Discrete Ph	ase Domain		
methyl-alcohol-liquid 🔹	linear	<b>v</b>		<ul> <li>none</li> </ul>		•	
Evaporating Species	Devolatilizing Species	Product	Species				
ch3oh 🔹		-		*			
Point Properties Physical	Models Turbulent Dispersio	on Parce	Wet Combustio	on Compon	ents UDF	Multiple Reaction	ons
Particle Rotation	Rough Wall Model     Breakup     Enable Breakup						
Enable Rotation	Chaple preakup						
Enable Rotation	Breakun Model						
Enable Rotation	Breakup Model	-					
Enable Rotation	ТАВ	-					
Enable Rotation	(	•					

- i. In the **Breakup** group, ensure that **Enable Breakup** is selected and **TAB** is selected from the **Breakup Model** drop-down list.
- ii. Retain the default values of 0 for **y0** and 2 for **Breakup Parcels**.
- iii. In the Drag Parameters group box, select dynamic-drag from the Drag Law drop-down list.

The dynamic-drag law is available only when the Breakup model is used.

q. In the **Turbulent Dispersion** tab, define the turbulent dispersion.

Injection Name		Injection Ty	pe			Number of Strea
injection-0		air-blast-ato	mizer		•	600
Particle Type Massless O Inert O Drop	let 🔿 Combusting 🛇 Multi	component	Laws Custom			
Material	Diameter Distribution	Oxidizing S	pecies	Discrete Phase Do	main	
methyl-alcohol-liquid 🔹	linear	¥]	~	none		•
Evaporating Species ch3oh •	Devolatilizing Species	Product S	pecies *			
Point Properties Physical	Models Turbulent Disper	sion Parcel	Wet Combustion	Components	UDF	Multiple Reaction
Stochastic Tracking Stochastic Tracking Stochastic Tracking Stochastic Tracking Random Eddy Lifetime Number of Tries Time Scale Constant 0.15	Cloud Tracking Cloud Model Min. Cloud Diameter ( 0 Max. Cloud Diameter 100000					
	0	K File C	ancel Help			

i. Enable **Discrete Random Walk Model** and **Random Eddy Lifetime** in the **Stochastic Tracking** group box.

These models will account for the turbulent dispersion of the droplets.

ii. Click **OK** to close the **Set Injection Properties** dialog box.

Note

To modify the existing injection, select its name in the **Injections** list and click **Set...**, or simply double-click the injection of interest.

r. Close the **Injections** dialog box.

#### Note

In the case that the spray injection would be striking a wall, you should specify the wall boundary conditions for the droplets. Though this tutorial does have wall zones, they are a part of the atomizer apparatus. You need not change the wall boundary conditions any further because these walls are not in the path of the spray droplets.

- s. Click OK to close the Discrete Phase Model dialog box.
- 2. Specify the droplet material properties.

**The Setup**  $\rightarrow$  **Create/Edit...** 

When secondary atomization models (such as **Breakup**) are used, several droplet properties need to be specified.

Create/Edit Materials		<b>—</b>
Name	Material Type	Order Materials by
methyl-alcohol-liquid	droplet-particle	Name
Chemical Formula	Fluent Droplet Particle Materials	Chemical Formula
ch3oh <l></l>	methyl-alcohol-liquid (ch3oh <l>)</l>	Fluent Database
	Mixture	
	none	User-Defined Database
Properties		
Saturation Vapor Pressure (pascal) piecewise-lin	near Edit)	
Heat of Pyrolysis (j/kg) constant	▼)[Edit]	
Temperature Averaging Coefficient none	•)[Edit]	
Composition Averaging Coefficient	▼ [Edit]	
	····· ································	
Chan	ge/Create Delete Close Help	4

- a. Ensure droplet-particle is selected in the Material Type drop-down list.
- b. Enter 0.0056 kg/m-s for Viscosity in the Properties group box.
- c. Ensure that piecewise-linear is selected from the Saturation Vapor Pressure drop-down list.

Scroll down to find the **Saturation Vapor Pressure** drop-down list.

- d. Click the **Edit...** button next to **Saturation Vapor Pressure** to open the **Piecewise-Linear Profile** dialog box.
- e. Retain the default values and click OK to close the Piecewise-Linear Profile dialog box.
- f. Select convection/diffusion-controlled from the Vaporization Model drop-down list.
- g. Click OK to close the Convection/Diffusion Model dialog box.
- h. Click **Change/Create** to accept the change in properties for the methanol droplet material and close the **Create/Edit Materials** dialog box.

## 16.4.9. Solution

1. Increase the under-relaxation factor for **Discrete Phase Sources**.

Solving  $\rightarrow$  Controls  $\rightarrow$  Controls...

Solution Controls
-------------------

Turbulent Viscosity		_
1		
:h3oh		
0.75		
02		
0.75		
inergy		- 11
0.75		 E
Discrete Phase Sources		
0.9		 ٦ 🛛
		-
efault		
quations	Advanced	
Set All Species URFs Tog		

In the **Pseudo Transient Explicit Relaxation Factors** group box, change the under-relaxation factor for **Discrete Phase Sources** to 0.9.

2. Remove the convergence criteria.

Residual Monitors				<b>X</b>
Options  Print to Console  Plot  Window  1  Curves  Axes  Iterations to Plot  1000	Equations Residual continuity x-velocity y-velocity z-velocity	Monitor		
Iterations to Store	Residual Values           Normalize           Scale           Compute Local	Scale	Iterations	Convergence Criterion
OK Plot	Renormalize		Cancel Hel	p

**Solving**  $\rightarrow$  **Reports**  $\rightarrow$  **Residuals...** 

- a. Select **none** from the **Convergence Criterion** drop-down list.
- b. Click **OK** to close the **Residual Monitors** dialog box.
- 3. Enable the plotting of mass fraction of **ch3oh**.

**Solving**  $\rightarrow$  Reports  $\rightarrow$  Definitions  $\rightarrow$  New  $\rightarrow$  Surface Report  $\rightarrow$  Mass-Weighted Average...

Name       Report Type         ch3oh_outlet       Mass-Weighted Average       •         Options       Custom Vectors       •         Per Surface       dpm-mean-velocity       •         Average Over       Custom Vectors       •         1       •       Field Variable         Report Files [0/0]       •       •       •         Mass fraction of ch3oh       •       •         Mass fraction of ch3oh       •       •         Report Plots [0/0]       •       •       •         Report Plots [0/0]       •       •       •         Image: The state of the sta
Options   Per Surface   Average Over   1   Field Variable Species Mass fraction of ch3oh Surfaces Fiter Text angle=15 atomizer-wall central_air co-flow-air default-interior fluid
Per Surface   Average Over   1   Image: Species in the second s
Per Surface   Average Over   1   Field Variable     Report Files [0/0]     Species     Mass fraction of ch3oh     Surfaces Filter Text     angle=15   atomizer-wall   central_air   co-flow-air   default-interior   fluid
Average Over          1       Custom Vectors         1       Field Variable         Report Files [0/0]       Species         Variable       Mass fraction of ch3oh         Surfaces       Fiter Text         angle=15         atomizer-wall         central_air         co-flow-air         default-interior         fluid
1   Report Files [0/0]     Field Variable     Species     Mass fraction of ch3oh     Surfaces Filter Text     angle=15   atomizer-wall   central_air   co-flow-air   default-interior   fluid
Report Files [0/0]       Species         Mass fraction of ch3oh         Surfaces Filter Text         angle=15         atomizer-wall         central_air         co-flow-air         default-interior         fluid
Report Files [0/0]       Species         Mass fraction of ch3oh         Surfaces Filter Text         angle=15         atomizer-wall         central_air         co-flow-air         default-interior         fluid
Report Plots [0/0]       Image:
Report Plots [0/0]       Image:
Report Plots [0/0]
Report Plots [0/0]
Report Plots [0/0]
co-flow-air default-interior fluid
default-interior fluid
fluid
outer-wall
outlet
Create periodic-a
Report File periodic-b
suiding siz
Report Plot
Frequency 1
Print to Console
Create Output Parameter New Surface 💌
OK Compute Cancel Help

- a. Enter ch3oh\_outlet for Name of the surface report definition.
- b. In the Create group box, enable Report Plot and Print to Console.
- c. Select Species... and Mass fraction of ch3oh from the Field Variable drop-down lists.
- d. Select **outlet** from the **Surfaces** selection list.
- e. Click **OK** to save the surface report definition settings and close the **Surface Report Definition** dialog box.

Fluent automatically generates the **ch3oh\_outlet-rplot** report plot under the **Solution/Monit-ors/Report Plots** tree branch.

4. Enable the plotting of the sum of the **DPM Mass Source**.

Volume Report Definition		
Name		Report Type
dpm-mass-source		Sum 🔻
Options		Field Variable
		Discrete Phase Sources
Per Zone		DPM Mass Source 👻
Average Over		
1		Cell Zones Filter Text
Report Files [0/0]	<b>x</b> - <b>-</b>	fluid
Report Plots [0/0]	x	
Create	,	
Report File		
Report Plot		
Frequency 1		
Print to Console		
Create Output Parameter		
	OK (	Compute Cancel Help

**Solving**  $\rightarrow$  Reports  $\rightarrow$  Definitions  $\rightarrow$  New  $\rightarrow$  Volume Report  $\rightarrow$  Sum...

- a. Enter **dpm-mass-source** for **Name**.
- b. In the Create group box, enable Report Plot and Print to Console.
- c. Select Discrete Phase Sources... and DPM Mass Source from the Field Variable drop-down lists.
- d. Select **fluid** from the **Cell Zones** selection list.
- e. Click **OK** to save the volume report definition settings and close the **Volume Report Definition** dialog box.

Fluent automatically generates the **dpm-mass-source-rplot** report plot under **Solution/Monit-ors/Report Plots** tree branch.

f. Modify the attributes of the **dpm-mass-source-rplot** report plot axes.

-	۴	
<b>Solution</b> $\rightarrow$ Monitors $\rightarrow$ Report Plots $\rightarrow$ dpm-mass-source-rplot	ц,	Edit

Edit Report Plot	
Name	
dpm-mass-source-rplot	
Available Report Definitions [0/0]	Selected Report Definitions [0/1]
	dpm-mass-source
	Add>> < <remove< td=""></remove<>
Options	New - Edit
Plot Window 3  Curves Axes	
Get Data Every 1 🜩 Iteration	
Plot Title dpm-mass-source-rplot	
X-Axis Label iteration	
Y-Axis Label Sum of dpm-mass-source	
Print to Console	
0	K Cancel Help

i. In the **Plot Window** group box, click the **Axes...** button to open the **Axes** dialog box.

Axes - Report Plots		<b>X</b>
Axis X Y Label	Number Format Type exponential Precision 2	Major Rules Color foreground Weight 1
Options Log Auto Range Major Rules Minor Rules	Range Minimum O Maximum O	Minor Rules Color dark gray * Weight 1
Apply	Close Help	

- ii. Select **Y** in the **Axis** list.
- iii. Select exponential from the Type drop-down list.
- iv. Set Precision to 2.
- v. Click Apply and close the Axes dialog box.
- vi. Click OK to close the Edit Report Plot dialog box.
- 5. Create a DPM report definition for tracking the total mass present in the domain.

### **Solving** $\rightarrow$ Reports $\rightarrow$ Definitions $\rightarrow$ New $\rightarrow$ DPM Report $\rightarrow$ Mass in Domain...

DPM Report Definition		<b>×</b>	
Name		Output Quantity	
dpm-mass-in-domain		Mass in Domain 🔹	
Average Over		Injections [1/1]	
Report Files [0/0]	<b>x</b> -	injection-0	
Report Plots [0/1]	<b></b>		
dpm-mass-source-rplot			
Create		1	
Report File			
Report Plot			
Frequency 1			
Print to Console			
Create Output Parameter		Show Mass Flow / Change Rate	
OK Compute Cancel Help			

- a. Enter dpm-mass-in-domain for Name.
- b. In the Create group box, enable Report Plot and Print to Console.
- c. Select injection-0 from the Injections selection list.
- d. Clear Show Mass Flow / Change Rate.
- e. Click **OK** to save the volume report definition settings and close the **DPM Report Definition** dialog box.

Fluent automatically generates the **dpm-mass-in-domain-rplot** report plot under **Solution/Mon-itors/Report Plots** tree branch.

f. Modify the attributes of the **dpm-mass-in-domain-rplot** report plot axes (in a manner similar to that for the **dpm-mass-source-rplot** plot).

**□** Solution → Monitors → Report Plots → dpm-mass-in-domain-rplot  $\stackrel{\bigcirc}{\Box}$  Edit...

- i. In the **Plot Window** group box, click the **Axes...** button to open the **Axes** dialog box.
- ii. Select **Y** in the **Axis** list.
- iii. Select **exponential** from the **Type** drop-down list.
- iv. Set Precision to 2.
- v. Click **Apply** and close the **Axes** dialog box.
- vi. Click OK to close the Edit Report Plot dialog box.
- 6. Create a DPM report definition for tracking the mass of the evaporated particles.

### Solving $\rightarrow$ Reports $\rightarrow$ Definitions $\rightarrow$ New $\rightarrow$ DPM Report $\rightarrow$ Evaporated Mass...

DPM Report Definition	
Name	Output Quantity
dpm-evaporated-mass	Evaporated Mass
Average Over	
1	Injections [1/1]
Report Files [0/0]	injection-0
Report Plots [0/1]	
dpm-mass-source-rplot	
Create	
Report File	
Report Plot	
Frequency 1 🚔	
Print to Console	
Create Output Parameter	Show Mass Flow / Change Rate
OK Compute	Cancel Help

- a. Enter **dpm-evaporated-mass** for **Name**.
- b. In the Create group box, enable Report Plot and Print to Console.
- c. Select injection-0 from the Injections selection list.
- d. Ensure that the Show Mass Flow / Change Rate option is selected.
- e. Click **OK** to save the volume report definition settings and close the **DPM Report Definition** dialog box.

Fluent automatically generates the **dpm-evaporated-mass-rplot** report plot under **Solution/Mon-itors/Report Plots** tree branch.

- f. Modify the attributes of the **dpm-evaporated-mass-rplot** report plot axes in a manner similar to that for the **dpm-mass-source-rplot** plot.
- 7. Request 300 more iterations (Figure 16.6: Convergence History of Mass Fraction of ch3oh on Fluid (p. 699), Figure 16.7: Convergence History of DPM Mass Source on Fluid (p. 700), Figure 16.8: Convergence History of Total Mass in Domain (p. 700), and Figure 16.9: Convergence History of Evaporated Particle Mass (p. 701)).

**Solving**  $\rightarrow$  Run Calculation

It can be concluded that the solution is converged because the number of particle tracks are constant and the flow variable plots are flat.

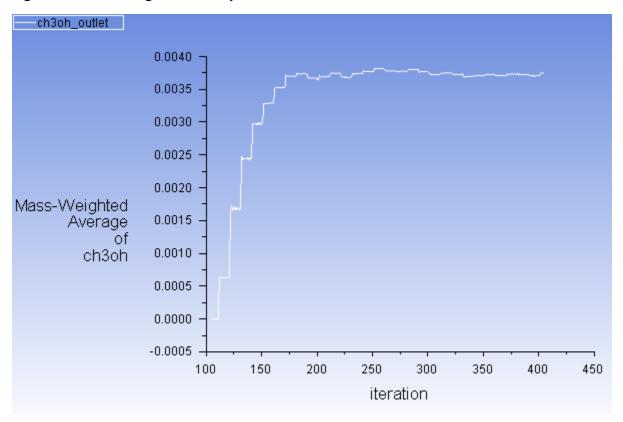


Figure 16.6: Convergence History of Mass Fraction of ch3oh on Fluid



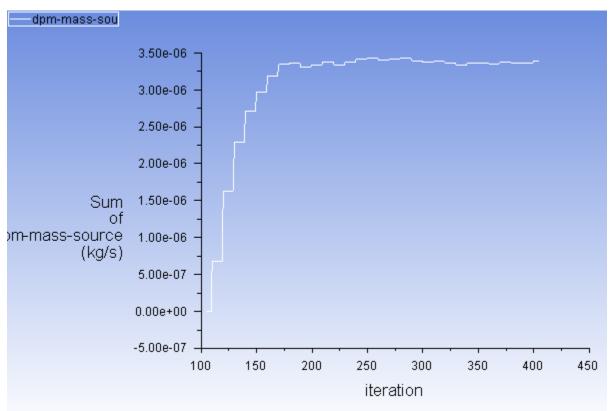
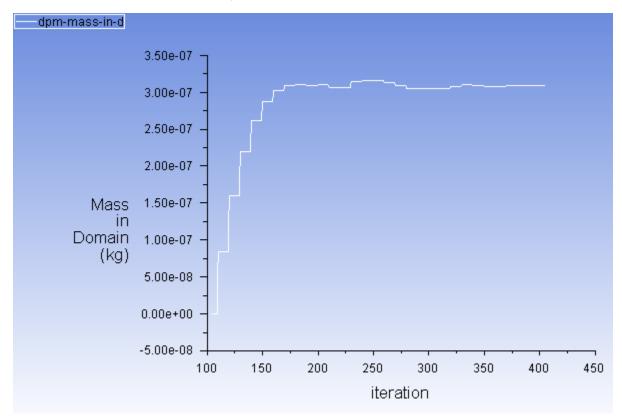


Figure 16.8: Convergence History of Total Mass in Domain



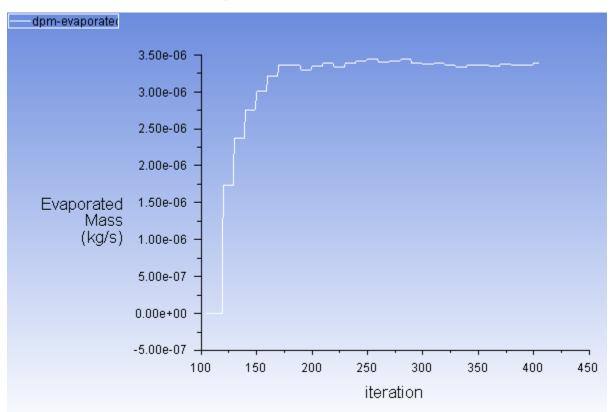


Figure 16.9: Convergence History of Evaporated Particle Mass

8. Save the case and data files (spray2.cas.gz and spray2.dat.gz).

File → Write → Case & Data...

## 16.4.10. Postprocessing

1. Display the trajectories of the droplets in the spray injection (Figure 16.10: Particle Tracks for the Spray Injection (p. 703)).

This will allow you to review the location of the droplets.

Postprocessing  $\rightarrow$  Graphics  $\rightarrow$  Particle Tracks  $\rightarrow$  Edit...

Particle Tracks		
Vode Values	Track Style     point     Attributes	Color by Particle Variables  Particle Diameter
XY Plot  Write to File Filter  Filter by	Vector Style          none         Attributes         Pulse Mode         Ocontinuous         Single	Min (m)       Max (m)         8.155428e-06       8.700575e-05         Update Min/Max         Track Single Particle Stream         Stream ID Skip       Coarsen         1       0       1         1       0       1
Reporting Report Type Off Summary Current Position	Report to File Console Display Pulse Track	Release from Injections [1/1]

a. Enable Draw Mesh in the Options group box.

The **Mesh Display** dialog box will open.

- i. Retain the current display settings.
- ii. Close the **Mesh Display** dialog box.
- b. Retain the default selection of **point** from the **Track Style** drop-down list.
- c. Select Particle Variables... and Particle Diameter from the Color by drop-down lists.

This will display the location of the droplets colored by their diameters.

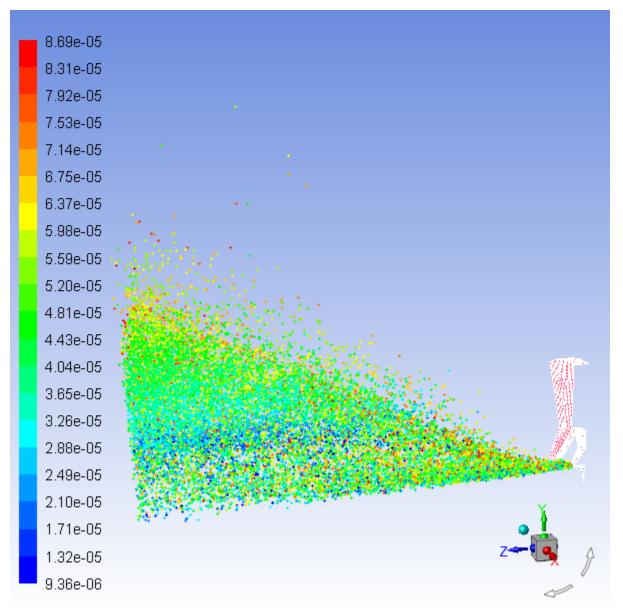
- d. Select injection-0 from the Release from Injections selection list.
- e. Click **Display**. As an optional exercise, you can increase the particle size by clicking the **Attributes...** button in the **Particle Tracks** dialog box and adjusting the **Marker Size** value in the **Track Style Attributes** dialog box.
- f. Close the **Particle Tracks** dialog box.
- g. Restore the 30-degree section to obtain the view as shown in Figure 16.10: Particle Tracks for the Spray Injection (p. 703).

 $\overset{\text{\tiny Were}}{=} Viewing \rightarrow Display \rightarrow Views...$ 

- i. Click the **Define...** button to open the **Graphics Periodicity** dialog box.
- ii. Click Reset and close the Graphics Periodicity dialog box.

- iii. Close the **Views** dialog box.
- h. Use the mouse to obtain the view shown in Figure 16.10: Particle Tracks for the Spray Injection (p. 703).

Figure 16.10: Particle Tracks for the Spray Injection



The air-blast atomizer model assumes that a cylindrical liquid sheet exits the atomizer, which then disintegrates into ligaments and droplets. Appropriately, the model determines that the droplets should be input into the domain in a ring. The radius of this disk is determined from the inner and outer radii of the injector.

### Note

The maximum diameter of the droplets is about  $10^{-4}$  m or 0.1 mm. This is slightly smaller than the film height. The inner diameter and outer diameter of the injector are 3.5 mm and 4.5 mm, respectively. Hence the film height is 0.5 mm. The range in the droplet sizes

is due to the fact that the air-blast atomizer automatically uses a distribution of droplet sizes.

Also note that the droplets are placed a slight distance away from the injector. Once the droplets are injected into the domain, their behavior will be determined by secondary models. For instance, they may collide/coalesce with other droplets depending on the secondary models employed. However, once a droplet has been introduced into the domain, the air-blast atomizer model no longer affects the droplet.

2. Display the mean particle temperature field (Figure 16.11: Contours of DPM Temperature (p. 705)).

Contours				×
Options  Filled  Global Range  Auto Range  Clip to Range  Draw Profiles	Contours of Discrete Phase Variab DPM Temperature Min (k) 260	Max (k) 271.6142		•
Draw Mesh	Surfaces Filter Text		-0	<b>Z</b> - <b>X</b>
Coloring Banded Smooth Levels Setup 20 1 =	angle=15 atomizer-wall central_air co-flow-air default-interior fluid outer-wall New Surface ▼	Close Hel		
			_	

Postprocessing  $\rightarrow$  Graphics  $\rightarrow$  Contours  $\rightarrow$  Edit...

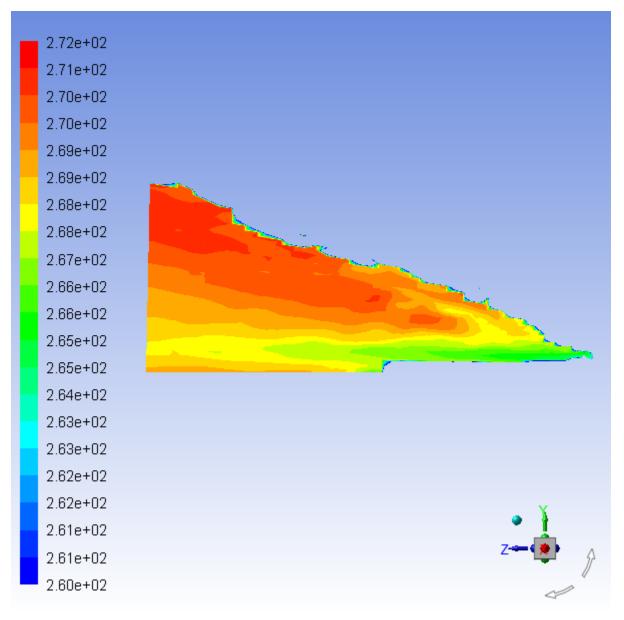
- a. Ensure that Filled is enabled in the Options group box
- b. Disable Draw Mesh.
- c. Select Discrete Phase Variables... and DPM Temperature from the Contours of drop-down lists.
- d. Disable Auto Range.

The Clip to Range option will automatically be enabled.

- e. Click Compute to update the Min and Max fields.
- f. Enter 260 for Min.
- g. Select angle=15 from the Surfaces selection list.

- h. Click **Display** and close the **Contours** dialog box.
- i. Use the mouse to obtain the view shown in Figure 16.11: Contours of DPM Temperature (p. 705).

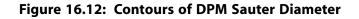
Figure 16.11: Contours of DPM Temperature

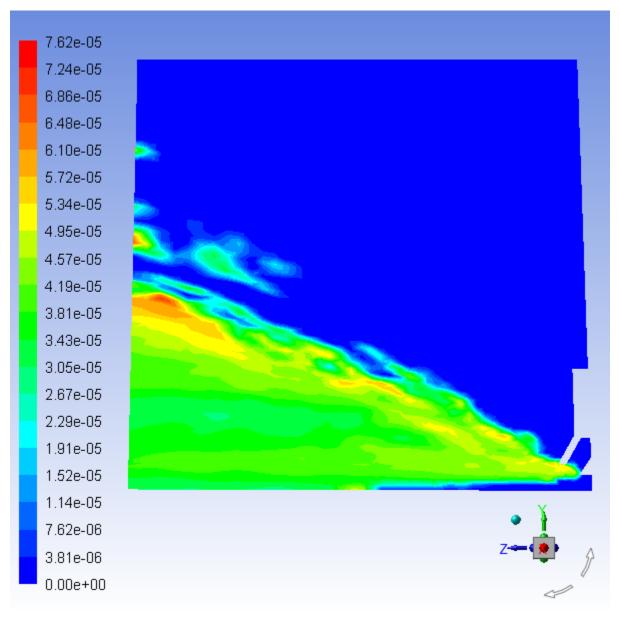


3. Display the mean Sauter diameter (Figure 16.12: Contours of DPM Sauter Diameter (p. 706)).

Postprocessing  $\rightarrow$  Graphics  $\rightarrow$  Contours  $\rightarrow$  Edit...

- a. Enable Autorange in the Options group box.
- b. Select Discrete Phase Variables... and DPM D32 from the Contours of drop-down lists.
- c. Select angle=15 from the Surfaces selection list.
- d. Click **Display** and close the **Contours** dialog box.





4. Display vectors of DPM mean velocity colored by DPM velocity magnitude (Figure 16.13: Vectors of DPM Mean Velocity Colored by DPM Velocity Magnitude (p. 708)).

Postprocessing  $\rightarrow$  Graphics  $\rightarrow$  Vectors  $\rightarrow$  Edit...

<b>Vectors</b>		<b>—X</b> —	
Options	Vectors of		
🗹 Global Range	dpm-mean-velocity 🔹		
Auto Range	Color by		
Clip to Range	Discrete Phase Variables		
Auto Scale Draw Mesh	DPM Velocity Magnitude 🗸		
	Min (m/s)	Max (m/s)	
Style	0	75.4233	
arrow 🔻	Surfaces Filter Text	x	
7 0 🚖	angle=15	×	
Vector Options	atomizer-wall	=	
Custom Vectors	central_air		
custom vectors	co-flow-air		
	default-interior fluid		
		•	
	New Surface 🔻		
	Display Compute	Close Help	

- a. Select **dpm-mean-velocity** from the **Vectors of** drop-down lists.
- b. Select **Discrete Phase Variables...** and **DPM Velocity Magnitude** from the **Color by** drop-down lists.
- c. Enter 7 for **Scale**.
- d. Select angle=15 from the Surfaces selection list.
- e. Click **Display** and close the **Contours** dialog box.

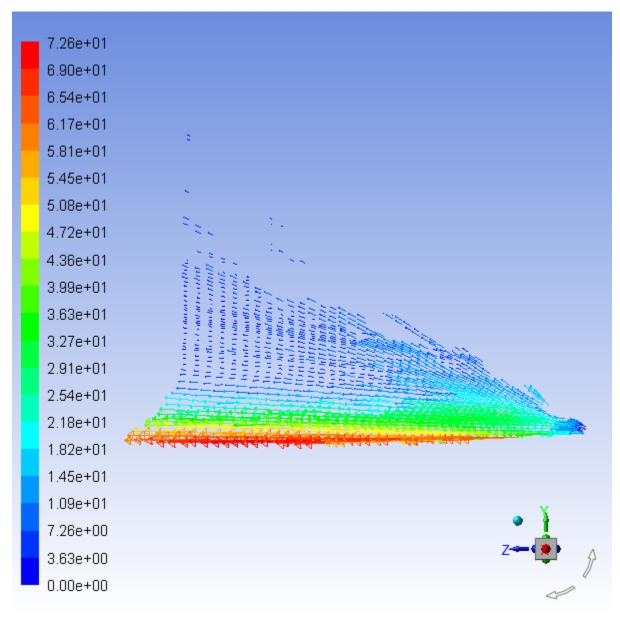


Figure 16.13: Vectors of DPM Mean Velocity Colored by DPM Velocity Magnitude

5. Create an isosurface of the methanol mass fraction.

Postprocessing  $\rightarrow$  Surface  $\rightarrow$  Create  $\rightarrow$  Iso-Surface...

Iso-Surface			<b>×</b>
Surface of Constant Species	•	From Surface Filter Text	x. <u>.</u>
Mass fraction of ch3oh	۱ <b>-</b>		*
Min Max		atomizer-wall central_air	
0	0.009669546	co-flow-air	
Iso-Values		default-interior	
0.002		fluid	*
< □	•	From Zones Fiter Text	x
New Surface Name		fluid	
methanol-mf=0.002			
	Create Compute	Manage Close Help	.H

- a. Select Species... and Mass fraction of ch3oh from the Surface of Constant drop-down lists.
- b. Click **Compute** to update the minimum and maximum values.
- c. Enter 0.002 for Iso-Values.
- d. Enter methanol-mf=0.002 for the New Surface Name.
- e. Click **Create** and then close the **Iso-Surface** dialog box.
- 6. Display the isosurface you just created (**methanol-mf=0.002**).
  - Postprocessing  $\rightarrow$  Graphics  $\rightarrow$  Mesh  $\rightarrow$  Edit...

💶 Mesh Display	,		
Options Nodes	Edge Type All	Surfaces Filter Text	
Edges	Feature	default-interior	
Faces	Outline	fluid	
Partitions		methanol-mf=0.002	
Overset		outer-wall	
Shrink Factor F	Feature Angle	outlet	
		penodic-a	
0	20	periodic-b	
Outline	Interior	swirling_air	
Adjacency		New Surface 🔻	
Display Colors Close Help			

a. Deselect atomizer-wall and select methanol-mf=0.002 in the Surfaces selection list.

b. Click the Colors... button to open the Mesh Colors dialog box.

Mesh Colors		<b>—</b>		
Options <ul> <li>Color by Type</li> <li>Color by ID</li> </ul> Sample	Types interior outlet periodic rans-les-interface symmetry axis wall free-surface internal traction interface surface	Colors          background         black         blue         cyan         dark blue         dark gray         dark green         dark red         foreground         green         light blue         ight gray		
Reset Colors Close Help				

i. Select surface in the Types list and green in the Colors list.

Scroll down the **Types** list to locate **surface**. The isosurface will now be displayed in green, which contrasts better with the rest of the mesh.

- ii. Close the **Mesh Colors** dialog box.
- c. Click **Display** in the **Mesh Display** dialog box.

The graphics display will be updated to show the isosurface.

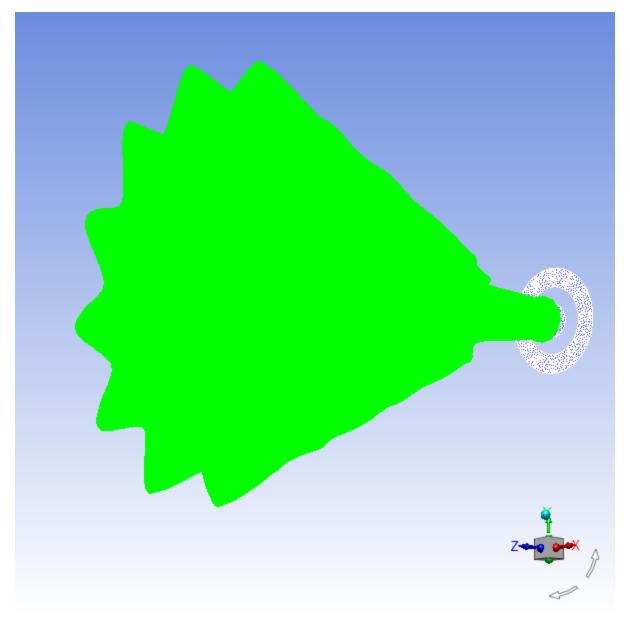
- 7. Modify the view to include the entire atomizer.

  - a. Click Define... to open the Graphics Periodicity dialog box.

Craphics Period	icity			
Cell Zones Filter T	ext 🐻 🗖	Associated Surfaces Filter Text		
fluid		angle=15		
		atomizer-wall		
		central_air		
		co-flow-air default-interior		
		fluid		
Deriedic Ture	Axis Direction			
Periodic Type		Axis Origin		
<ul> <li>Translational</li> <li>Detectional</li> </ul>	X (m) 0	X (m) 0		
Rotational	Y (m) 0	Y (m) 0		
Angle (deg)	Z (m) 1	Z (m) 0		
30				
Number of Repeats 12 文				
	Set	Reset Default Close Help		

- i. Select **fluid** from the **Cell Zones** list.
- ii. Ensure that **Rotational** is selected from the **Periodic Type** list and the **Number of Repeats** is set to 12.
- iii. Click Set and close the Graphics Periodicity dialog box.
- b. Click **Apply** and close the **Views** dialog box.
- c. Click **Display** and close the **Mesh Display** dialog box.
- d. Use the mouse to obtain the view shown in Figure 16.14: Full Atomizer Display with Surface of Constant Methanol Mass Fraction (p. 711).

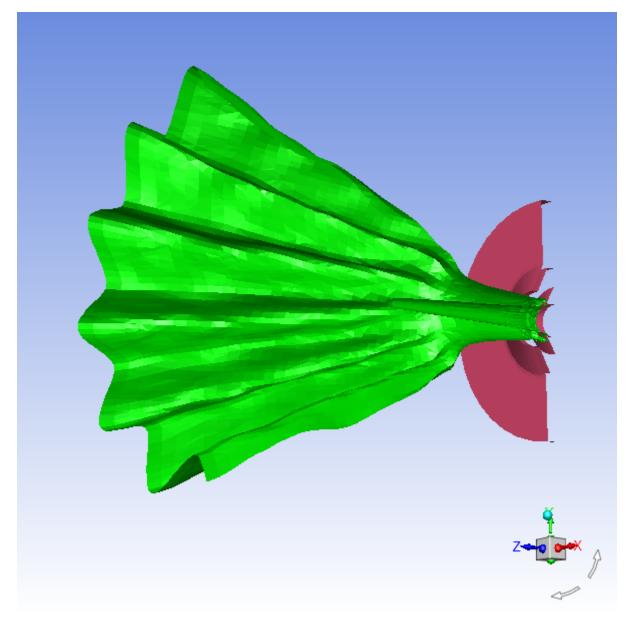




e. This view can be improved to resemble Figure 16.15: Atomizer Display with Surface of Constant Methanol Mass Fraction Enhanced (p. 712) by changing some of the following variables:

- Disable Edges in the Mesh Display dialog box
- Select only atomizer-wall and methanol-mf=0.002 in the Surfaces list of the Mesh Display dialog box
- Change the Number of Repeats to 6 in the Graphics Periodicity dialog box
- Enable Lighting and change it to Flat in the Viewing tab (Display group)
- Enable Headlight check in the Viewing tab (Display group)

#### Figure 16.15: Atomizer Display with Surface of Constant Methanol Mass Fraction Enhanced



8. Save the case and data files (spray3.cas.gz and spray3.dat.gz).



## 16.5. Summary

In this tutorial, a spray injection was defined for an air-blast atomizer and the solution was calculated using the discrete phase model in ANSYS Fluent. The location of methanol droplet particles after exiting the atomizer and an isosurface of the methanol mass fraction were examined.

## 16.6. Further Improvements

This tutorial guides you through the steps to reach an initial solution. You may be able to obtain a more accurate solution by using an appropriate higher-order discretization scheme and by adapting the mesh. Mesh adaption can also ensure that the solution is independent of the mesh. These steps are demonstrated in Introduction to Using ANSYS Fluent: Fluid Flow and Heat Transfer in a Mixing Elbow (p. 121).

# **Chapter 17: Using the VOF Model**

This tutorial is divided into the following sections:

17.1. Introduction17.2. Prerequisites17.3. Problem Description17.4. Setup and Solution17.5. Summary17.6. Further Improvements

# 17.1. Introduction

This tutorial examines the flow of ink as it is ejected from the nozzle of a printhead in an inkjet printer. Using ANSYS Fluent's volume of fluid (VOF) multiphase modeling capability, you will be able to predict the shape and motion of the resulting droplets in an air chamber.

This tutorial demonstrates how to do the following:

- Set up and solve a transient problem using the pressure-based solver and VOF model.
- Copy material from the property database.
- Define time-dependent boundary conditions with a user-defined function (UDF).
- Patch initial conditions in a subset of the domain.
- Automatically save data files at defined points during the solution.
- Examine the flow and interface of the two fluids using volume fraction contours.

# 17.2. Prerequisites

This tutorial is written with the assumption that you have completed one or more of the introductory tutorials found in this manual:

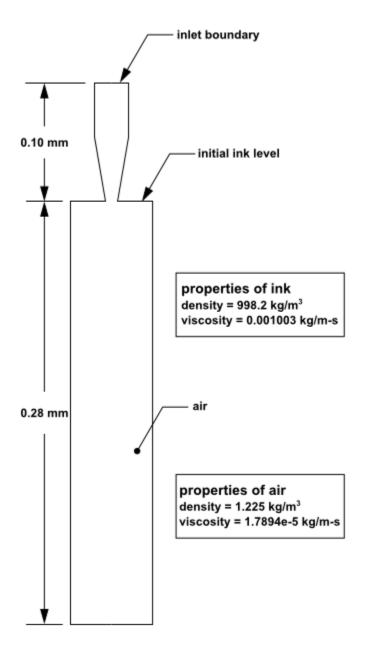
- Introduction to Using ANSYS Fluent in ANSYS Workbench: Fluid Flow and Heat Transfer in a Mixing Elbow (p. 1)
- Parametric Analysis in ANSYS Workbench Using ANSYS Fluent (p. 73)
- Introduction to Using ANSYS Fluent: Fluid Flow and Heat Transfer in a Mixing Elbow (p. 121)

and that you are familiar with the ANSYS Fluent tree and ribbon structure. Some steps in the setup and solution procedure will not be shown explicitly.

# **17.3. Problem Description**

The problem considers the transient tracking of a liquid-gas interface in the geometry shown in Figure 17.1: Schematic of the Problem (p. 716). The axial symmetry of the problem enables a 2D geometry to be used. The computation mesh consists of 24,600 cells. The domain consists of two regions: an ink chamber and an air chamber. The dimensions are summarized in Table 17.1: Ink Chamber Dimensions (p. 716).

### Figure 17.1: Schematic of the Problem



#### Table 17.1: Ink Chamber Dimensions

Ink Chamber, Cylindrical Region: Radius (mm)	0.015
Ink Chamber, Cylindrical Region: Length (mm)	0.050
Ink Chamber, Tapered Region: Final Radius (mm)	0.009

Ink Chamber, Tapered Region: Length (mm)	0.050
Air Chamber: Radius (mm)	0.030
Air Chamber: Length (mm)	0.280

The following is the chronology of events modeled in this simulation:

- At time zero, the nozzle is filled with ink, while the rest of the domain is filled with air. Both fluids are assumed to be at rest. To initiate the ejection, the ink velocity at the inlet boundary (which is modeled in this simulation by a user-defined function) suddenly increases from 0 to 3.58 m/s and then decreases according to a cosine law.
- After 10 microseconds, the velocity returns to zero.

The calculation is run for 30 microseconds overall, that is, three times longer than the duration of the initial impulse.

Because the dimensions are small, the double-precision version of ANSYS Fluent will be used. Air will be designated as the primary phase, and ink (which will be modeled with the properties of liquid water) will be designated as the secondary phase. Patching will be required to fill the ink chamber with the secondary phase. Gravity will not be included in the simulation. To capture the capillary effect of the ejected ink, the surface tension and prescription of the wetting angle will be specified. The surface inside the nozzle will be modeled as neutrally wettable, while the surface surrounding the nozzle orifice will be non-wettable.

## 17.4. Setup and Solution

The following sections describe the setup and solution steps for this tutorial:

17.4.1. Preparation
17.4.2. Reading and Manipulating the Mesh
17.4.3. General Settings
17.4.4. Models
17.4.5. Materials
17.4.6. Phases
17.4.7. Operating Conditions
17.4.8. User-Defined Function (UDF)
17.4.9. Boundary Conditions
17.4.10. Solution
17.4.11. Postprocessing

## 17.4.1. Preparation

To prepare for running this tutorial:

- 1. Set up a working folder on the computer you will be using.
- 2. Go to the ANSYS Customer Portal, https://support.ansys.com/training.

#### Note

If you do not have a login, you can request one by clicking **Customer Registration** on the log in page.

- 3. Enter the name of this tutorial into the search bar.
- 4. Narrow the results by using the filter on the left side of the page.
  - a. Click ANSYS Fluent under Product.
  - b. Click **18.0** under **Version**.
- 5. Select this tutorial from the list.
- 6. Click the **vof\_R180.zip** link to download the input files.
- 7. Unzip  $vof_R180.zip$  file you downloaded to your working folder.

The files inkjet.msh and inlet1.c can be found in the vof directory created on unzipping the file.

8. Use Fluent Launcher to start the **2D** version of ANSYS Fluent.

Fluent Launcher displays your **Display Options** preferences from the previous session.

- 9. Ensure that the **Display Mesh After Reading** and **Workbench Color Scheme** options are enabled.
- 10. Enable **Double-Precision**.

For more information about the Fluent Launcher, see starting ANSYS Fluent using the Fluent Launcher in the Fluent Getting Started Guide.

#### Note

The double precision solver is recommended for modeling multiphase flows simulation.

11. Ensure that the **Serial** processing option is selected.

# 17.4.2. Reading and Manipulating the Mesh

1. Read the mesh file inkjet.msh.

### **File** $\rightarrow$ Read $\rightarrow$ Mesh...

A warning message will be displayed twice in the console. You need not take any action at this point, as the issue will be resolved when you define the solver settings in General Settings (p. 723).

2. Examine the mesh (Figure 17.2: Default Display of the Nozzle Mesh (p. 719)).

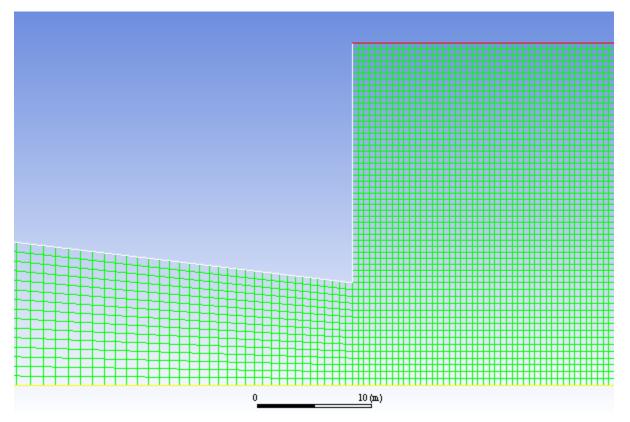


-			
	0	100 (m.)	

### Tip

By zooming in with the middle mouse button, you can see that the interior of the model is composed of a fine mesh of quadrilateral cells (see Figure 17.3: The Quadrilateral Mesh (p. 720)).

### Figure 17.3: The Quadrilateral Mesh



3. Set graphics display options



Display Options	
Rendering	Graphics Window
Line Width 1	Active Window Close
Point Symbol (+)	1 🗘 Set
Animation Option	Color Scheme
All	Workbench -
☑ Double Buffering	
Outer Face Culling	Lighting Attributes
Hidden Line Removal	Lights On
Hidden Surface Removal	Lighting Gouraud 🔻
Removal Method	
Hardware Z-buffer 👻	Layout
Display Timeout	Titles
Timeout in seconds 60	Axes
	Ruler
	Logo
	Color
	White 🔻
	Colormap
	Colormap Alignment
	Left 🔻
Apply Info Lights	Close Help

a. Ensure that **All** is selected from the **Animation Option** drop-down list.

Selecting **All** will allow you to see the movement of the entire mesh as you manipulate the **Camera** view in the next step.

- 4. Click **Apply** and close the **Display Options** dialog box.
- 5. Manipulate the mesh display to show the full chamber upright.

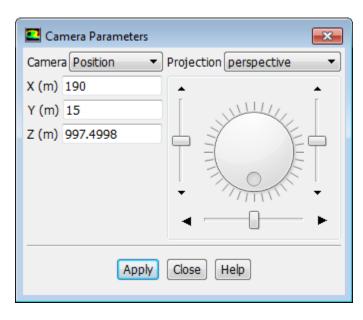


<b>2</b> Views		<b>X</b>
Views back front Save Name front	Actions Default Auto Scale Previous Save Delete Read Write	Mirror Planes [1/1]
Appl	y Camera	Close Help

- a. Select **front** from the **Views** selection list.
- b. Select axis from the Mirror Planes selection list.
- c. Click Apply.

The mesh display is updated to show both sides of the chamber.

d. Click the Camera... button to open the Camera Parameters dialog box.



### Note

You may notice that the scale of the dimensions in the Camera Parameters dialog box appear very large given the problem dimensions. This is because you have not yet scaled the mesh to the correct units. You will do this in a later step. i. Drag the indicator of the dial with the left mouse button in the clockwise direction until the upright view is displayed (Figure 17.4: Mesh Display of the Nozzle Mirrored and Upright (p. 723)).

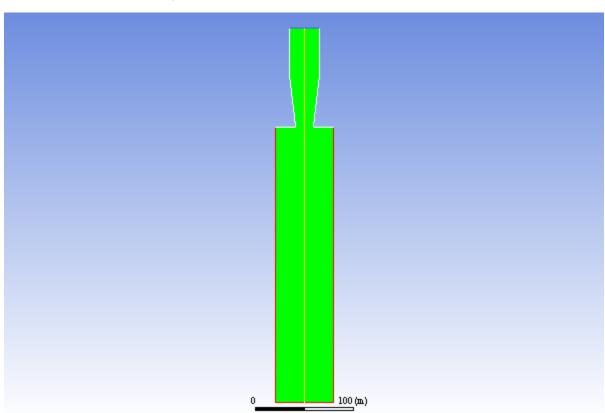


Figure 17.4: Mesh Display of the Nozzle Mirrored and Upright

- ii. Close the **Camera Parameters** dialog box.
- e. Close the Views dialog box.

## 17.4.3. General Settings

1. Check the mesh.



ANSYS Fluent will perform various checks on the mesh and report the progress in the console. Make sure that the reported minimum volume is a positive number.

2. Scale the mesh.



💶 Scale M	esh			
-Domain E	Extents			Scaling
Xmin (m)	0	Xmax (m)	0.0003799999	Convert Units
Ymin (m)	0	Ymax (m)	3e-05	Specify Scaling Factors
				Mesh Was Created In
				<select> 🔻</select>
View Leng	th Unit In			Scaling Factors
m				X 1e-6
				Y 1e-6
Scale Unscale				
Close Help				

- a. Select **Specify Scaling Factors** from the **Scaling** group box.
- b. Enter 1e-6 for **X** and **Y** in the **Scaling Factors** group box.
- c. Click Scale and close the Scale Mesh dialog box.
- 3. Check the mesh.

**Setting Up Domain**  $\rightarrow$  Mesh  $\rightarrow$  Check

### Note

It is a good idea to check the mesh after you manipulate it (that is, scale, convert to polyhedra, merge, separate, fuse, add zones, or smooth and swap.) This will ensure that the quality of the mesh has not been compromised.

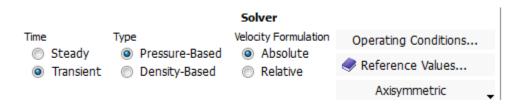
4. Define the units for the mesh.



Set Units			
Quantities		Units	Set All to
spring-constant spring-constant-angular stefan-boltzmann-constant surface-density	*	n/m lbf/ft dyn/cm	default si british
surface-tension surface-tension-gradient temperature temperature-difference			cgs
temperature-inverse thermal-conductivity	Ŧ	Factor 0.001 Offset 0	
	New List	Close Help	.H.

- a. Select length from the Quantities list.
- b. Select **mm** from the **Units** list.
- c. Select surface-tension from the Quantities list.
- d. Select dyn/cm from the Units list.
- e. Close the Set Units dialog box.
- 5. Retain the default setting of **Pressure-Based** in the **Solver** group box of the **Setting Up Physics** ribbon tab.

Setting Up Physics  $\rightarrow$  Solver

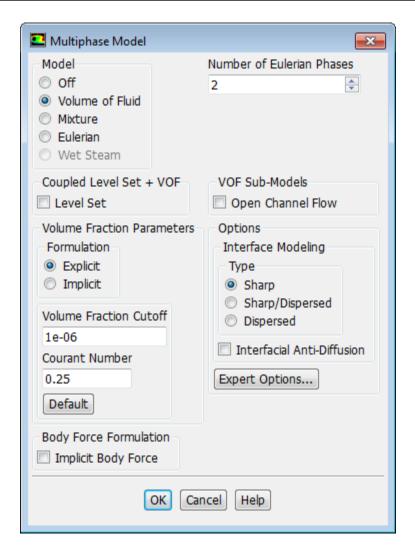


- 6. Select Transient from the Time list.
- 7. Select Axisymmetric from the drop-down list in the Solver group box (below Reference Values).

## 17.4.4. Models

1. Enable the Volume of Fluid multiphase model.

```
Setting Up Physics \rightarrow Models \rightarrow Multiphase...
```



a. Select Volume of Fluid from the Model list.

The **Multiphase Model** dialog box expands to show related inputs.

b. Retain the default settings and click **OK** to close the **Multiphase Model** dialog box.

### 17.4.5. Materials

The default properties of air and water defined in ANSYS Fluent are suitable for this problem. In this step, you will make sure that both materials are available for selection in later steps.

1. Add water to the list of fluid materials by copying it from the ANSYS Fluent materials database.

Setting Up Physics  $\rightarrow$  Materials  $\rightarrow$  Create/Edit...

Create/Edit Materials			×
Name		Material Type	Order Materials by
water-liquid		fluid	<ul> <li>Name</li> </ul>
Chemical Formula		Fluent Fluid Materials	Chemical Formula
h2o <l></l>		water-liquid (h2o <l>)</l>	Fluent Database
		Mixture	
		none	User-Defined Database
Viscosity (kg/m-s)	998.2	Edit Edit	
		Change/Create Delete Close Help	

a. Click **Fluent Database...** in the **Create/Edit Materials** dialog box to open the **Fluent Database Materials** dialog box.

Fluent Database Materials		
Fluent Database Materials		×.
Fluent Fluid Materials [1/563]	<b> </b>	Material Type fluid
vinyl-silylidene (h2cchsih) vinyl-trichlorosilane (sicl3ch2ch) vinylidene-chloride (ch2ccl2)	•	<ul> <li>Order Materials by</li> <li>Name</li> <li>Chemical Formula</li> </ul>
water-liquid (h2o <l>)</l>		
water-vapor (h2o)		
wood-volatiles (wood_vol)	-	
Copy Materials from Case Delete		
Properties		
Density (kg/m3)	constant	✓ View ▲
	998.2	=
Cp (Specific Heat) (j/kg-k)	constant	▼ [View]
	4182	
Thermal Conductivity (w/m-k)	constant	▼ View
	0.6	
Viscosity (kg/m-s)	constant	▼ View
	0.001003	-
New Edit	Save Copy Clo	se Help

i. Select water-liquid (h2o < l >) from the Fluent Fluid Materials selection list.

Scroll down the Fluent Fluid Materials list to locate water-liquid (h2o < l >).

- ii. Click **Copy** to copy the information for water to your list of fluid materials.
- iii. Close the Fluent Database Materials dialog box.
- b. Click Change/Create and close the Create/Edit Materials dialog box.

### 17.4.6. Phases

In the following steps, you will define water as the secondary phase. When you define the initial solution, you will patch water in the nozzle region. In general, you can specify the primary and secondary phases whichever way you prefer. It is a good idea to consider how your choice will affect the ease of problem setup, especially with more complicated problems.

Setting Up Physics  $\rightarrow$  Phases  $\rightarrow$  List/Show All...

Phases	
Phases	
air - Primary Phase	
water-liquid - Secondary Phase	
Edit Interaction ID 3	
Close Help	

1. Specify air (air) as the primary phase.

In the **Phases** dialog box, select **phase 1 – Primary Phase** and click **Edit...** to open the **Primary Phase** dialog box.

💶 Primary Phase		×
Name		
air		
Phase Material air	▼ Edit	
	OK Delete Cancel Help	

- a. Enter air for Name.
- b. Retain the default selection of air in the Phase Material drop-down list.
- c. Click OK to close the Primary Phase dialog box.
- 2. Specify water (water-liquid) as the secondary phase.

In the **Phases** dialog box, select **phase 2 – Secondary Phase** and click **Edit...** to open the **Secondary Phase** dialog box.

Secondary Phase	×
Name	
water-liquid	
Phase Material water-liquid	
OK Delete Cancel Help	

- a. Enter water-liquid for Name.
- b. Select water-liquid from the Phase Material drop-down list.
- c. Click **OK** to close the **Secondary Phase** dialog box.
- 3. Specify the interphase interaction.

In the **Phases** dialog box, click **Interaction...** to open the **Phase Interaction** dialog box.

Phase Interaction						×
ispersion Turbulen	ce Interaction Collis	ions Slip	Heat Mass	Reactions	Surface Tension	<b>₫</b> ₿¢
<ul> <li>Surface Tension For Model</li> <li>Continuum Surface</li> <li>Continuum Surface</li> <li>Surface Tension Coeff</li> </ul>	Adhesion E Force Vall Ad E Stress Jump A	hesion				
water-liquid	air		73.5		▼ Edit	
		OK Can	cel Help			

- a. Click the **Surface Tension** tab.
- b. Enable Surface Tension Force Modeling.

The surface tension inputs is displayed and the **Continuum Surface Force** model is set as the default.

- c. Enable Wall Adhesion so that contact angles can be prescribed.
- d. Select constant from the Surface Tension Coefficient drop-down list.
- e. Enter 73.5 dyn/cm for Surface Tension Coefficient.
- f. Click **OK** to close the **Phase Interaction** dialog box.
- 4. Close the **Phases** dialog box.

## 17.4.7. Operating Conditions

1. Set the operating reference pressure location.

**E** Setup  $\rightarrow$  **C** Boundary Conditions  $\rightarrow$  Operating Conditions...

Operating Conditions	<b>—</b>
Pressure	Gravity
Operating Pressure (pascal)	Gravity
101325 P	
Reference Pressure Location	
X (mm) 0.10	
Y (mm) 0.03	
Z (mm) 0	
OK Cancel Help	

You will set the **Reference Pressure Location** to be a point where the fluid will always be 100 % air.

- a. Enter 0.10 mm for X.
- b. Enter 0.03 mm for Y.
- c. Click OK to close the Operating Conditions dialog box.

## 17.4.8. User-Defined Function (UDF)

1. Interpret the UDF source file for the ink velocity distribution (inlet1.c).

## 

Interpreted UDFs
Source File Name
inlet1.c Browse
CPP Command Name
срр
Stack Size
10000 🚖
Display Assembly Listing
Use Contributed CPP
Interpret Close Help

a. Enter inlet1.c for Source File Name.

If the UDF source file is not in your working directory, then you must enter the entire directory path for **Source File Name** instead of just entering the file name. Alternatively, click the **Browse...** button and select **inlet1.c** in the vof directory that was created after you unzipped the original file.

b. Click Interpret.

The UDF defined in inlet1.c is now visible and available for selection as **udf membrane\_speed** in the drop-down lists of relevant graphical user interface dialog boxes.

c. Close the Interpreted UDFs dialog box.

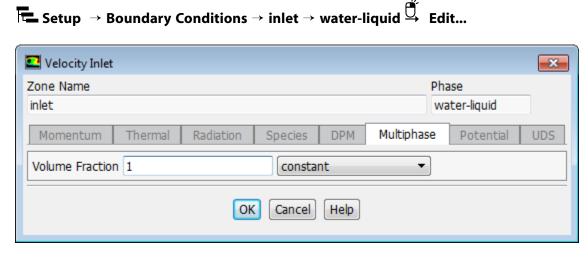
# 17.4.9. Boundary Conditions

1. Set the boundary conditions at the inlet (**inlet**) for the mixture by selecting **mixture** from the **Phase** dropdown list in the **Boundary Conditions** task page.

**E** Setup  $\rightarrow$  Boundary Conditions  $\rightarrow$  inlet  $\stackrel{\square}{\stackrel{\smile}{\rightarrow}}$  Edit...

<b>C</b> Velocity Inlet					×	
Zone Name inlet				nase hixture		
Momentum	Thermal Radiation	Species DPN	1 Multiphase	Potential	UDS	
Velocit	y Specification Method Mag	initude, Normal	to Boundary		•	
	Reference Frame Abs	olute			•	
	Velocity Magnitude (m/s) udf membrane_speed					
Supersonic/Init	Supersonic/Initial Gauge Pressure (pascal) 0 constant					
	ОК	Cancel He	þ			

- a. Select **udf membrane\_speed** from the **Velocity Magnitude** drop-down list.
- b. Click **OK** to close the **Velocity Inlet** dialog box.
- 2. Set the boundary conditions at the inlet (**inlet**) for the secondary phase by selecting **water-liquid** from the **Phase** drop-down list in the **Boundary Conditions** task page.



a. Click the **Multiphase** tab and enter 1 for the **Volume Fraction**.

- b. Click **OK** to close the **Velocity Inlet** dialog box.
- 3. Set the boundary conditions at the outlet (**outlet**) for the secondary phase by selecting **water-liquid** from the **Phase** drop-down list in the **Boundary Conditions** task page.

**Example :** Setup  $\rightarrow$  Boundary Conditions  $\rightarrow$  outlet  $\rightarrow$  water-liquid  $\stackrel{\square}{\rightarrow}$  Edit...

Pressure Outlet	<b>—</b>
Zone Name	Phase
outlet	water-liquid
Momentum Thermal Radiation Species DPM	Multiphase Potential UDS
Volume Fraction Specification Method Backflow Volume Fraction	•
Backflow Volume Fraction 0	constant 🔹
OK Cancel Help	

- a. Click the **Multiphase** tab and retain the default setting of **0** for the **Backflow Volume Fraction**.
- b. Click **OK** to close the **Pressure Outlet** dialog box.
- 4. Set the conditions at the top wall of the air chamber (**wall\_no\_wet**) for the mixture by selecting **mixture** from the **Phase** drop-down list in the **Boundary Conditions** task page.



🞴 Wall					×
Zone Name			Phase		
wall_no_wet			mixtur	e	
Adjacent Cell Zone					
fluid					
Momentum Thermal	Radiation Species	DPM Mul	ltiphase U	DS Wall Film	Potential
	lotion Relative to Adjacent Cell :	Zone			
Shear Condition <ul> <li>No Slip</li> <li>Specified Shear</li> <li>Specularity Coefficien</li> <li>Marangoni Stress</li> </ul>	ıt				
Wall Roughness					
Roughness Height (mm)	0	constant	~		
Roughness Constant	0.5	constant	~	]	
Wall Adhesion Contact Angles (deg)					
water-liquid	air	:	175	constant	
	ОК	Cancel Help			

- a. Enter 175 degrees for Contact Angles.
- b. Click **OK** to close the **Wall** dialog box.

#### Note

This angle affects the dynamics of droplet formation. You can repeat this simulation to find out how the result changes when the wall is hydrophilic (that is, using a small contact angle, say 10 degrees).

5. Set the conditions at the side wall of the ink chamber (**wall\_wet**) for the mixture.



🖸 Wall						×	
			Dha				
Zone Name				Phase			
wall_wet			mix	cure			
Adjacent Cell Zone							
Momentum Thermal Rad	diation Species	DPM N	Iultiphase	UDS	Wall Film	Potential	
Wall Motion Motion Stationary Wall Relative Moving Wall	ve to Adjacent Cell 2	Zone					
Shear Condition <ul> <li>No Slip</li> <li>Specified Shear</li> <li>Specularity Coefficient</li> <li>Marangoni Stress</li> </ul>							
Wall Roughness							
Roughness Height (mm) 0		constant		-			
Roughness Constant 0.5		constant		-			
Wall Adhesion Contact Angles (deg)							
water-liquid	air		90	consta	ant	•	
	ОК	ancel Hel	p			H.	

- a. Retain the default setting of **90** degrees for **Contact Angles**.
- b. Click **OK** to close the **Wall** dialog box.

## 17.4.10. Solution

1. Set the solution methods.



#### Solution Methods

Pressure-Velocity Coupling
Scheme
Fractional Step 🔹
Spatial Discretization
Gradient
Least Squares Cell Based 🔹
Pressure
PRESTO!
Momentum
QUICK 🔻
Volume Fraction
Compressive 🔻
Transient Formulation
First Order Implicit
Non-Iterative Time Advancement
Frozen Flux Formulation
Warped-Face Gradient Correction
High Order Term Relaxation Options
Default

a. Enable Non-Iterative Time Advancement.

The non-iterative time advancement (NITA) scheme is often advantageous compared to the iterative schemes as it is less CPU intensive. Although smaller time steps must be used with NITA compared to the iterative schemes, the total CPU expense is often smaller. If the NITA scheme leads to convergence difficulties, then the iterative schemes (for example, PISO, SIMPLE) should be used instead.

- b. Select **Fractional Step** from the **Scheme** drop-down list in the **Pressure-Velocity Coupling** group box.
- c. Retain the default selection of **Least Squares Cell Based** from the **Gradient** drop-down list in the **Spatial Discretization** group box.
- d. Retain the default selection of **PRESTO!** from the **Pressure** drop-down list.
- e. Select QUICK from the Momentum drop-down list.
- f. Select **Compressive** from the **Volume-Fraction** drop-down list.
- 2. Enable the plotting of residuals during the calculation.

**Solving**  $\rightarrow$  Reports  $\rightarrow$  Residuals...

Residual Monitors					×
Options	Equations				
Print to Console	Residual	Monitor	Check Converge	ence Absolute Criteria	
Plot	continuity	<b>V</b>	$\checkmark$	0.001	
Window	x-velocity	<b>V</b>	$\checkmark$	0.001	
1 Curves Axes	y-velocity		$\checkmark$	0.001	
Iterations to Plot	Residual Values				
Iterations to Store	Normalize		Iterations 5	Convergence Conditio	ns
1000	Scale				
	Compute Loca	l Scale			
ОК	Plot Renormaliz	e Car	ncel Help		.4

- a. Ensure **Plot** is selected in the **Options** group box.
- b. Click **OK** to close the **Residual Monitors** dialog box.
- 3. Initialize the solution after reviewing the default initial values.

Solving  $\rightarrow$  Initialization  $\rightarrow$  Options...

Solution Initialization
Initialization Methods
O Hybrid Initialization
Standard Initialization
Compute from
•
Reference Frame
Relative to Cell Zone
O Absolute
Initial Values
Gauge Pressure (pascal)
0
Axial Velocity (m/s)
0
Radial Velocity (m/s)
0
water-liquid Volume Fraction
0
Initialize Reset Patch
Reset DPM Sources Reset Statistics

- a. Retain the default settings for all the parameters and click **Initialize** (either in the ribbon or in the **Solution Initialization** task page.
- 4. Define a register for the ink chamber region.

**EXAMPLE** Setting Up Domain  $\rightarrow$  Adapt  $\rightarrow$  Mark/Adapt Cells  $\rightarrow$  Region...

Region Adaption				
Options	Input Coordinate			
Inside	X Min (mm)	X Max (mm)		
Outside	0	0.1		
Shapes	Y Min (mm)	Y Max (mm)		
Quad	0	0.03		
O Circle	Z Min (mm)	Z Max (mm)		
Ocylinder	0	0		
Manage	0			
Controls	Colort Deinte with Maure			
Controls Select Points with Mouse				
Adapt Mark Close Help				

- a. Retain the default setting of **0** mm for **X Min** and **Y Min** in the **Input Coordinates** group box.
- b. Enter 0.10 mm for **X Max**.
- c. Enter 0.03 mm for Y Max.
- d. Click Mark.

ANSYS Fluent will report in the console that 1500 cells were marked for refinement while zero cells were marked for coarsening.

#### Tip

You can display and manipulate adaption registers, which are generated using the **Mark** command, using the **Manage Adaption Registers** dialog box. Click the **Manage...** button in the **Region Adaption** dialog box to open the **Manage Adaption Registers** dialog box. For details, see Adapting the Mesh (p. 174).

- e. Close the Region Adaption dialog box.
- 5. Patch the initial distribution of the secondary phase (water-liquid).

Solving  $\rightarrow$  Initialization  $\rightarrow$  Patch...

I Patch			<b>×</b>
Reference Frame Relative to Cell Zone Absolute Phase water-liquid Variable	Value 1 Use Field Function Field Function	Zones to Patch Filter Text	
Volume Fraction Volume Fraction Patch Options Patch Reconstructed Interface Volumetric Smoothing		Registers to Patch [1/1]	
	Patch Clo	se Help	h.

- a. Select water-liquid from the Phase drop-down list.
- b. Select Volume Fraction from the Variable list.
- c. Enter 1 for Value.
- d. Select hexahedron-r0 from the Registers to Patch selection list.
- e. Click Patch and close the Patch dialog box.
- 6. Request the saving of data files every 200 steps.

**Solving**  $\rightarrow$  Activities  $\rightarrow$  Autosave...

Autosave
Save Data File Every (Time Steps) 200 두
Data File Quantities Save Associated Case Files <ul> <li>Only if Modified</li> <li>Each Time</li> </ul>
File Storage Options Retain Only the Most Recent Files Maximum Number of Data Files 0
File Name
inkjet Browse
Append File Name with time-step 🔻
OK Cancel Help

- a. Enter 200 for Save Data File Every (Time Steps).
- b. Ensure that time-step is selected from the Append File Name with drop-down list.
- c. Enter inkjet for the File Name.

ANSYS Fluent will append the time step value to the file name prefix (inkjet). The standard .dat extension will also be appended. This will yield file names of the form inkjet-1-00200.dat, where 200 is the time step number.

Optionally, you can add the extension .gz to the end of the file name (for example, inkjet.gz), which instructs ANSYS Fluent to save the data files in a compressed format, yielding file names of the form inkjet-1-00200.dat.gz.

- d. Click **OK** to close the **Autosave** dialog box.
- 7. Save the initial case file (inkjet.cas.gz).

File  $\rightarrow$  Write  $\rightarrow$  Case...

8. Run the calculation.

	1000C [H	Solving	$\rightarrow$	Run	Calcul	ation
--	----------	---------	---------------	-----	--------	-------

	Run Calculation	
Preview Mesh Motion	Time Step Size (s) 1e-08 P	
Input Summary	No. of Time Steps 3000 🌻	-/-
Advanced	Check Case	Calculate

a. Enter 1.0e-8 seconds for the **Time Step Size (s)**.

#### Note

Small time steps are required to capture the oscillation of the droplet interface and the associated high velocities. Failure to use sufficiently small time steps may cause differences in the results between platforms.

- b. Enter 3000 for the Number of Time Steps.
- c. Click Calculate.

The solution will run for 3000 iterations.

### 17.4.11. Postprocessing

1. Read the data file for the solution after 6 microseconds (inkjet-1-00600.dat.gz).

**File**  $\rightarrow$  Read  $\rightarrow$  Data...

2. Create and display a filled contour of water volume fraction after 6 microseconds (Figure 17.5: Contours of Water Volume Fraction After 6 μs (p. 744)).

Postprocessing  $\rightarrow$  Graphics  $\rightarrow$  Contours  $\rightarrow$  New...

Contours		
Contour Name		
contour-volume-fracti	on	
Options	Contours of	
Filled	Phases	
Node Values	Volume fraction 👻	
Global Range	Phase	
Auto Range Clip to Range	water-liquid 🔹	
Draw Profiles	Min Max	
Draw Mesh	0 0	
Coloring Banded Smooth Colormap Options	Surfaces Filter Text	
Save/Display Compute Close Help		

- a. Change the Contour Name to contour-volume-fraction.
- b. Enable **Filled** in the **Options** group box.
- c. Select Phases... and Volume fraction from the Contours of drop-down lists.
- d. Select water-liquid from the Phase drop-down list.
- e. Click Save/Display.

Tip

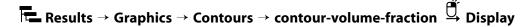
In order to display the contour plot in the graphics window, you may need to click the

Fit to Window button.

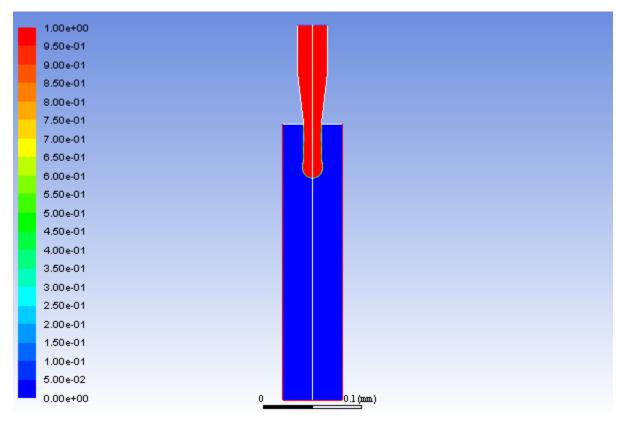
- Display contours of water volume fraction after 12, 18, 24, and 30 microseconds (Figure 17.6: Contours of Water Volume Fraction After 12 μs (p. 744) — Figure 17.9: Contours of Water Volume Fraction After 30 μs (p. 746)).
  - a. Read the data file for the solution after 12 microseconds (inkjet-1-01200.dat.gz).



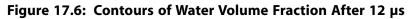
b. Reload the contour graphic saved in the previous step.

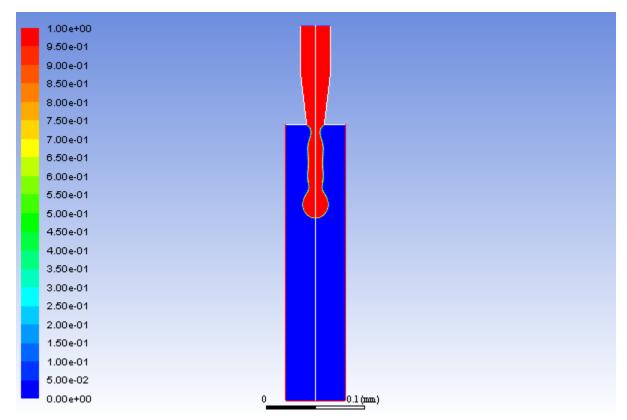


c. Repeat these steps for the 18, 24, and 30 microseconds files.



#### Figure 17.5: Contours of Water Volume Fraction After 6 µs





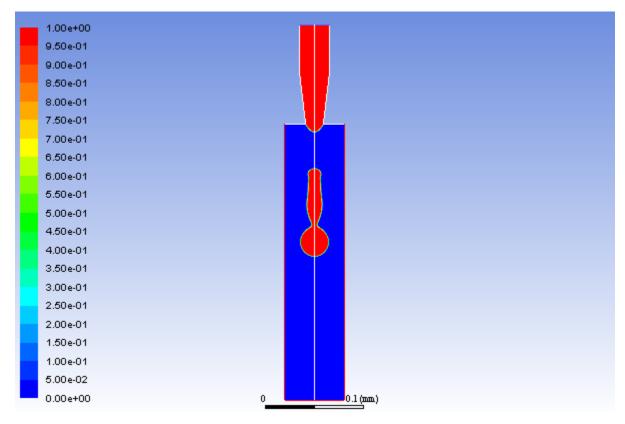
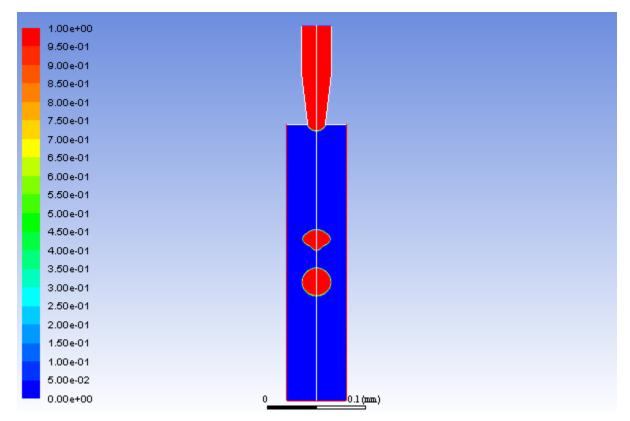


Figure 17.7: Contours of Water Volume Fraction After 18 µs





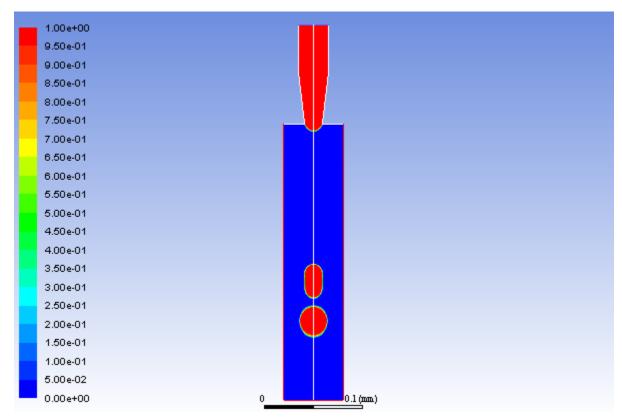


Figure 17.9: Contours of Water Volume Fraction After 30 µs

## 17.5. Summary

This tutorial demonstrated the application of the volume of fluid method with surface tension effects. The problem involved the 2D axisymmetric modeling of a transient liquid-gas interface, and postprocessing showed how the position and shape of the surface between the two immiscible fluids changed over time.

For additional details about VOF multiphase flow modeling, see the Fluent Theory Guide.

## **17.6. Further Improvements**

This tutorial guides you through the steps to reach an initial solution. You may be able to obtain a more accurate solution by using an appropriate higher-order discretization scheme and by adapting the mesh. Mesh adaption can also ensure that the solution is independent of the mesh. These steps are demonstrated in Introduction to Using ANSYS Fluent: Fluid Flow and Heat Transfer in a Mixing Elbow (p. 121).

# **Chapter 18: Modeling Cavitation**

This tutorial is divided into the following sections:

18.1. Introduction
18.2. Prerequisites
18.3. Problem Description
18.4. Setup and Solution
18.5. Summary
18.6. Further Improvements

# 18.1. Introduction

This tutorial examines the pressure-driven cavitating flow of water through a sharp-edged orifice. This is a typical configuration in fuel injectors, and brings a challenge to the physics and numerics of cavitation models because of the high pressure differentials involved and the high ratio of liquid to vapor density. Using the multiphase modeling capability of ANSYS Fluent, you will be able to predict the strong cavitation near the orifice after flow separation at a sharp edge.

This tutorial demonstrates how to do the following:

- Set boundary conditions for internal flow.
- Use the mixture model with cavitation effects.
- Calculate a solution using the pressure-based coupled solver.

# 18.2. Prerequisites

This tutorial is written with the assumption that you have completed one or more of the introductory tutorials found in this manual:

- Introduction to Using ANSYS Fluent in ANSYS Workbench: Fluid Flow and Heat Transfer in a Mixing Elbow (p. 1)
- Parametric Analysis in ANSYS Workbench Using ANSYS Fluent (p. 73)
- Introduction to Using ANSYS Fluent: Fluid Flow and Heat Transfer in a Mixing Elbow (p. 121)

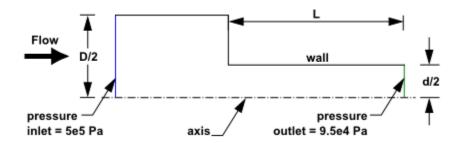
and that you are familiar with the ANSYS Fluent tree and ribbon structure. Some steps in the setup and solution procedure will not be shown explicitly.

# **18.3. Problem Description**

The problem considers the cavitation caused by the flow separation after a sharp-edged orifice. The flow is pressure driven, with an inlet pressure of  $5 \times 10^5$  Pa and an outlet pressure of  $9.5 \times 10^4$  Pa. The orifice diameter is  $4 \times 10^{-3}$  m, and the geometrical parameters of the orifice are D/d = 2.88 and L/d =

4, where D, d, and L are the inlet diameter, orifice diameter, and orifice length respectively. The geometry of the orifice is shown in Figure 18.1: Problem Schematic (p. 748).

#### Figure 18.1: Problem Schematic



# 18.4. Setup and Solution

The following sections describe the setup and solution steps for this tutorial: 18.4.1. Preparation

18.4.2. Reading and Checking the Mesh 18.4.3. Solver Settings 18.4.4. Models 18.4.5. Materials 18.4.6. Phases 18.4.7. Boundary Conditions 18.4.8. Operating Conditions 18.4.9. Solution 18.4.10. Postprocessing

## 18.4.1. Preparation

To prepare for running this tutorial:

- 1. Set up a working folder on the computer you will be using.
- 2. Go to the ANSYS Customer Portal, https://support.ansys.com/training.

#### Note

If you do not have a login, you can request one by clicking **Customer Registration** on the log in page.

- 3. Enter the name of this tutorial into the search bar.
- 4. Narrow the results by using the filter on the left side of the page.
  - a. Click ANSYS Fluent under Product.
  - b. Click **18.0** under **Version**.
- 5. Select this tutorial from the list.

- 6. Click the **cavitation\_R180.zip** link to download the input files.
- 7. Unzip the cavitation\_R180.zip file you downloaded to your working folder.

The mesh file cav.msh can be found in the cavitation directory created after unzipping the file.

8. Use the Fluent Launcher to start the **2D** version of ANSYS Fluent.

Fluent Launcher displays your **Display Options** preferences from the previous session.

For more information about Fluent Launcher, see starting ANSYS Fluent using the Fluent Launcher in the Fluent Getting Started Guide.

- 9. Ensure that the **Display Mesh After Reading** and **Workbench Color Scheme** options are enabled.
- 10. Ensure that the Serial processing option is selected.
- 11. Enable **Double Precision**.

#### Note

The double precision solver is recommended for modeling multiphase flows simulation.

### 18.4.2. Reading and Checking the Mesh

1. Read the mesh file cav.msh.

File  $\rightarrow$  Read  $\rightarrow$  Mesh...

As ANSYS Fluent reads the mesh file, it will report the progress in the console. You can disregard the warnings about the use of axis boundary conditions, as you will make the appropriate change to the solver settings in the next step.

2. Check the mesh.



ANSYS Fluent will perform various checks on the mesh and will report the progress in the console. Ensure that the reported minimum volume is a positive number.

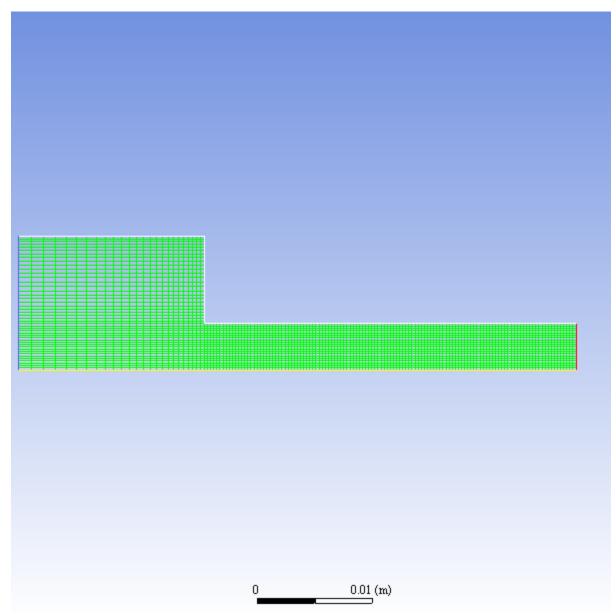
3. Check the mesh scale.

Setting Up Domain  $\rightarrow$  Mesh  $\rightarrow$  Transform  $\rightarrow$  Scale...

💶 Scale N	/lesh		<b>X</b>
Domain Ext	tents		Scaling
Xmin (m)	-0.016	Xmax (m) 0.032	<ul> <li>Convert Units</li> <li>Specify Scaling Factors</li> </ul>
Ymin (m)	0	Ymax (m) 0.01152	Mesh Was Created In <select></select>
View Lengt	h Unit In		Scaling Factors
			× 1
			Y 1
			Scale Unscale
		Close Help	

- a. Retain the default settings.
- b. Close the **Scale Mesh** dialog box.
- 4. Examine the mesh (Figure 18.2: The Mesh in the Orifice (p. 751)).





As seen in Figure 18.2: The Mesh in the Orifice (p. 751), half of the problem geometry is modeled, with an axis boundary (consisting of two separate lines) at the centerline. The quadrilateral mesh is slightly graded in the plenum to be finer toward the orifice. In the orifice, the mesh is uniform with aspect ratios close to 1, as the flow is expected to exhibit two-dimensional gradients.

When you display data graphically in a later step, you will mirror the view across the centerline to obtain a more realistic view of the model.

Since the bubbles are small and the flow is high speed, gravity effects can be neglected and the problem can be reduced to axisymmetrical. If gravity could not be neglected and the direction of gravity were not coincident with the geometrical axis of symmetry, you would have to solve a 3D problem.

### 18.4.3. Solver Settings

1. Specify an axisymmetric model.

#### General Mesh Scale... Check Report Quality Display... Solver Туре Velocity Formulation Pressure-Based Absolute Density-Based Relative Time 2D Space Steady Planar Transient Axisymmetric Axisymmetric Swirl Gravity Units... Help

a. Retain the default selection of Pressure-Based in the Type list.

The pressure-based solver must be used for multiphase calculations.

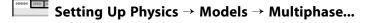
b. Select Axisymmetric in the 2D Space list.

#### Note

A computationally intensive, transient calculation is necessary to accurately simulate the irregular cyclic process of bubble formation, growth, filling by water jet re-entry, and break-off. In this tutorial, you will perform a steady-state calculation to simulate the presence of vapor in the separation region in the time-averaged flow.

### 18.4.4. Models

1. Enable the multiphase mixture model.



💶 Multiphase Model	<b>—</b>
Model Off Volume of Fluid Mixture Eulerian Wet Steam	Number of Eulerian Phases
Mixture Parameters	]
Body Force Formulation	]
ОКС	Cancel Help

a. Select **Mixture** in the **Model** list.

The *Multiphase Model* dialog box will expand.

b. Clear Slip Velocity in the Mixture Parameters group box.

In this flow, the high level of turbulence does not allow large bubble growth, so gravity is not important. It is also assumed that the bubbles have same velocity as the liquid. Therefore, there is no need to solve for the slip velocity.

- c. Click **OK** to close the **Multiphase Model** dialog box.
- 2. Enable the realizable k- $\varepsilon$  turbulence model with standard wall functions.

Setting Up Physics  $\rightarrow$  Models  $\rightarrow$  Viscous...

Viscous Model	<b>.</b>		
Model Laminar Spalart-Allmaras (1 eqn) k-epsilon (2 eqn) k-omega (2 eqn) Transition k-kl-omega (3 eqn) Transition SST (4 eqn) Reynolds Stress (5 eqn) Scale-Adaptive Simulation (SAS) Detached Eddy Simulation (DES)	Model Constants C2-Epsilon 1.9 TKE Prandtl Number 1 TDR Prandtl Number 1.2		
k-epsilon Model Standard RNG Realizable	User-Defined Functions Turbulent Viscosity None		
<ul> <li>Near-Wall Treatment</li> <li>Standard Wall Functions</li> <li>Scalable Wall Functions</li> <li>Non-Equilibrium Wall Functions</li> <li>Enhanced Wall Treatment</li> <li>Menter-Lechner</li> <li>User-Defined Wall Functions</li> </ul>			
Options Production Limiter			
OK Cancel Help			

- a. Select **k-epsilon (2 eqn)** in the **Model** list.
- b. Select Realizable in the k-epsilon Model list.
- c. Retain the default of Standard Wall Functions in the Near-Wall Treatment list.
- d. Click OK to close the Viscous Model dialog box.

### 18.4.5. Materials

For the purposes of this tutorial, you will be modeling the liquid and vapor phases as incompressible. Note that more comprehensive models are available for the densities of these phases, and could be used to more fully capture the affects of the pressure changes in this problem.

1. Create a new material to be used for the primary phase.

💶 Create/Edit Ma	terials			<b>—</b> ×
Name water		Material Type		Order Materials by
Chemical Formula		Fluent Fluid Materials		Chemical Formula
		water	•	
		Mixture		User-Defined Database
		none		·
Properties			_	
Density (kg/m3)	constant	▼ Edit		
	1000			
Viscosity (kg/m-s)	constant	▼ Edit		
	0.001		E	
			-	
			Ŧ	
	Change/Create	Delete Clos	se Help	

- a. Enter water for Name.
- b. Enter 1000 kg/m<sup>3</sup> for **Density**.
- c. Enter 0.001 kg/m-s for Viscosity.
- d. Click Change/Create.

A Question dialog box will open, asking if you want to overwrite air. Click Yes.

Question		×
?	Change/Create mixture and Overwrite air?	
	Yes No	

- 2. Copy water vapor from the materials database and modify the properties of your local copy.
  - a. In the **Create/Edit Materials** dialog box, click the **Fluent Database...** button to open the **Fluent Database Materials** dialog box.

Fluent Database Materials	
Fluent Fluid Materials [1/563]	Material Type [fluid
vinyl-silylidene (h2cchsih)         vinyl-trichlorosilane (sicl3ch2ch)         vinylidene-chloride (ch2ccl2)         water-liquid (h2o <l>)</l>	Order Materials by Name  Chemical Formula
water-vapor (h2o) wood-volatiles (wood_vol)	
Copy Materials from Case Delete Properties	I
Density (kg/m3) constant	▼ View ▲
0.5542 Cp (Specific Heat) (j/kg-k) piecewise-polynomial	▼ View
Thermal Conductivity (w/m-k) constant	▼ View
0.0261	
Viscosity (kg/m-s) constant	View
1.34e-05	
New Edit Save Copy Clo	se Help

i. Select water-vapor (h2o) from the Fluent Fluid Materials selection list.

Scroll down the list to find water-vapor (h2o).

ii. Click **Copy** to include water vapor in your model.

water-vapor appears under Fluid in the Materials task page

iii. Close the Fluent Database Materials dialog box.

Name		Material Trees	Order Materials by
water-vapor Chemical Formula		Material Type fluid	<ul> <li>Name</li> </ul>
		Fluent Fluid Materials	Chemical Formula
h2o		water-vapor (h2o)	Fluent Database
		Mixture	User-Defined Database
		none	<b>v</b>
Properties			
Density (kg/m3)	constant	- Edit	
	0.02558		
Viscosity (kg/m-s)	constant	▼ Edit	
	1.26e-06		
		E	
		-	

- b. Enter  $0.02558 \text{ kg/m}^3$  for **Density**.
- c. Enter 1.26e-06 kg/m-s for **Viscosity**.
- d. Click Change/Create and close the Create/Edit Materials dialog box.

### 18.4.6. Phases

**E** Setup  $\rightarrow$  Models  $\rightarrow$  Multiphase (Mixture)  $\rightarrow$  Phases  $\stackrel{\frown}{\sqcup}$  Edit...

Phases
Phases
phase-1 - Primary Phase phase-2 - Secondary Phase
phase-2 - Secondary Phase
Edit Interaction ID 2
Close Help

- 1. Specify liquid water as the primary phase.
  - a. In the **Phases** dialog box, select **phase 1 Primary Phase** and click **Edit...** to open the **Primary Phase** dialog box.

Primary Phase	×
Name	
liquid	
Phase Material water	
OK Cancel Help	

- b. Enter liquid for Name.
- c. Retain the default selection of water from the Phase Material drop-down list.
- d. Click OK to close the Primary Phase dialog box.
- 2. Specify water vapor as the secondary phase.
  - a. In the **Phases** dialog box, select **phase 2 Secondary Phase** and click **Edit...** to open the **Secondary Phase** dialog box.

Secondary Phase	×
Name	
vapor	
Phase Material water-vapor   Edit	
OK Cancel Help	

- b. Enter vapor for Name.
- c. Select water-vapor from the Phase Material drop-down list.
- d. Click **OK** to close the **Secondary Phase** dialog box.
- 3. Enable the cavitation model.
  - a. In the Phases dialog box, click Interaction... to open the Phase Interaction dialog box.

💶 Phi	ase Interact	tion												
Virtu	ual Mass	Drag	Lift	Wall Lub	rication	Turbulent Disp	ersion	Turbulen	ce Inte	eraction	Collisions	Slip	Heat	Mass
	oer of Mass Transfer	Transfe	r Med	hanisms 1	\$									
	From Phase			To Phase		Mechanism								i
1	liquid		•	vapor	•	cavitation			it					
										ОК	Cancel He	Þ		

b. In the Mass tab, set Number of Mass Transfer Mechanisms to 1.

The dialog box expands to show relevant modeling parameters.

- c. Click **OK** in the dialog box that appears.
- d. Ensure that **liquid** is selected from the **From Phase** drop-down list in the **Mass Transfer** group box.
- e. Select vapor from the To Phase drop-down list.
- f. Select cavitation from the Mechanism drop-down list.

The Cavitation Model dialog box will open to show the cavitation inputs.

Cavitation Model	<b>—</b>
<ul> <li>Model</li> <li>Schnerr-Sauer</li> <li>Zwart-Gerber-Belamri</li> </ul>	
Cavitation Properties	Model Constants
Vaporization Pressure: Pv (pascal)	Bubble Number Density
constant   Edit	1e+13
3540	
	Turbulence Factor
OK Cancel Help	]

- i. Retain the default settings.
- ii. Retain the value of 3540 Pa for Vaporization Pressure.

The vaporization pressure is a property of the working liquid, which depends mainly on the temperature and pressure. The default value is the vaporization pressure of water at 1 atmosphere and a temperature of 300 K.

- iii. Click **OK** to close the **Cavitation Model** dialog box.
- g. Click **OK** to close the **Phase Interaction** dialog box.
- 4. Close the **Phases** dialog box.

### **18.4.7. Boundary Conditions**

For the multiphase mixture model, you will specify conditions for the mixture (that is, conditions that apply to all phases) and the conditions that are specific to the primary and secondary phases. In this tutorial, boundary conditions are required only for the mixture and secondary phase of two boundaries: the pressure inlet (consisting of two boundary zones) and the pressure outlet. The pressure outlet is the downstream boundary, opposite the pressure inlets.



Boundary Conditions
Zone Filter Text
default-interior inlet_1 inlet_2 outlet symm_1 symm_2 wall
Phase Type ID mixture  pressure-inlet 8
EditCopyProfilesParametersOperating ConditionsDisplay MeshPeriodic Conditions
Help

1. Set the boundary conditions at **inlet\_1** for the mixture. Ensure that **mixture** is selected from the **Phase** drop-down list in the **Boundary Conditions** task page.



Pressure Inlet	:						×		
Zone Name					Pha	ase	_		
inlet_1					mi	xture			
Momentum	Thermal	Radiation	Species	DPM	Multiphase	Potential	UDS		
	Referen	ce Frame Ab	solute				•		
Gau	uge Total Pre	ssure (pascal	) 500000		const	ant			
Supersonic/Init	ial Gauge Pre	ssure (pascal	) 449000		const	ant			
Direction	Specification	n Method No	rmal to Bou	ndary			-		
Turbuler	ice								
	Specification	Method K ar	nd Epsilon				•		
Turbule	nt Kinetic En	ergy (m2/s2)	0.02		consta	ant	-		
Turbulent Dissipation Rate (m2/s3) 1 constant						-			
	OK Cancel Help								

- a. Enter 500000 Pa for Gauge Total Pressure.
- b. Enter 449000 Pa for Supersonic/Initial Gauge Pressure.

If you choose to initialize the solution based on the pressure-inlet conditions, the **Supersonic/Initial Gauge Pressure** will be used in conjunction with the specified stagnation pressure (the **Gauge Total Pressure**) to compute initial values according to the isentropic relations (for compressible flow) or Bernoulli's equation (for incompressible flow). Otherwise, in an incompressible flow calculation, ANSYS Fluent will ignore the **Supersonic/Initial Gauge Pressure** input.

- c. Retain the default selection of **Normal to Boundary** from the **Direction Specification Method** dropdown list.
- d. Select K and Epsilon from the Specification Method drop-down list in the Turbulence group box.
- e. Enter  $0.02 \text{ m}^2/\text{s}^2$  for **Turbulent Kinetic Energy**.
- f. Retain the value of  $1 \text{ m}^2/\text{s}^3$  for **Turbulent Dissipation Rate**.
- g. Click **OK** to close the **Pressure Inlet** dialog box.
- 2. Set the boundary conditions at **inlet-1** for the secondary phase.



- a. Select **vapor** from the **Phase** drop-down list.
- b. Click Edit... to open the Pressure Inlet dialog box.

Pressure	Inle	t						×
Zone Name							ase	_
inlet_1						va	por	
Momentu	m	Thermal	Radiation	Species	DPM	Multiphase	Potential	UDS
Volume Fra	ctio	n 0		constar	nt	•		
			ОК	Cancel	Help			

- i. In the Multiphase tab, retain the default value of 0 for Volume Fraction.
- ii. Click **OK** to close the **Pressure Inlet** dialog box.
- 3. Copy the boundary conditions defined for the first pressure inlet zone (**inlet\_1**) to the second pressure inlet zone (**inlet\_2**).



- a. Select mixture from the Phase drop-down list.
- b. Click **Copy...** to open the **Copy Conditions** dialog box.

Copy Conditions	<b>——</b>
From Boundary Zone Filter Text	o Boundary Zones Filter Text
	inlet_2
inlet_2 symm_1	E
symm_2	
·	•
Copy Close	Help

- i. Select inlet\_1 from the From Boundary Zone selection list.
- ii. Select inlet\_2 from the To Boundary Zones selection list.
- iii. Click **Copy**.

A **Question** dialog box will open, asking if you want to copy **inlet\_1** boundary conditions to **inlet\_2**. Click **OK**.

#### **Modeling Cavitation**

Question	
?	Copy inlet_1 boundary conditions to all the selected zones?
	OK Cancel

- iv. Close the **Copy Conditions** dialog box.
- 4. Set the boundary conditions at **outlet** for the mixture.

**E** Setup  $\rightarrow$  **Conditions**  $\rightarrow$  **E** outlet  $\rightarrow$  Edit...

Pressure Outlet									
Zone Name									
outlet	mixture								
Momentum	Thermal	Radiation	Species	DPM	Multiphase	Potential	UDS		
	Gaug	e Pressure (p	ascal) 9500	0		constant	•		
Backflow Dire	ction Specifi	cation Metho	d Normal to	Boundar	у		<b>•</b>		
- Turbulence -									
	Specific	ation Method	K and Epsil	on			<b></b>		
Backflow Tu	rbulent Kinet	ic Energy (m	2/s2) 0.02		C	onstant	•		
Backflow Turb	Backflow Turbulent Dissipation Rate (m2/s3) 1 constant								
OK Cancel Help									

- a. Enter 95000 *Pa* for **Gauge Pressure**.
- b. Select K and Epsilon from the Specification Method drop-down list in the Turbulence group box.
- c. Enter 0.02 m<sup>2</sup>/s<sup>2</sup> for **Backflow Turbulent Kinetic Energy**.
- d. Retain the value of  $1 \text{ m}^2/\text{s}^3$  for **Backflow Turbulent Dissipation Rate**.
- e. Click **OK** to close the **Pressure Outlet** dialog box.
- 5. Set the boundary conditions at **outlet** for the secondary phase.

**Setup**  $\rightarrow$  **\bigcirc Boundary Conditions**  $\rightarrow$  **\equiv outlet** 

- a. Select **vapor** from the **Phase** drop-down list.
- b. Click **Edit...** to open the **Pressure Outlet** dialog box.

Pressure Outlet	
Zone Name	Phase
outlet	vapor
Momentum Thermal Radiation Species DPM	Multiphase Potential UDS
Volume Fraction Specification Method Backflow Volume Fraction	ion 🔹
Backflow Volume Fraction 0	constant 💌
OK Cancel Help	

- i. In the Multiphase tab, retain the default value of 0 for Backflow Volume Fraction.
- ii. Click **OK** to close the **Pressure Outlet** dialog box.

### 18.4.8. Operating Conditions

1. Set the operating pressure.

Setup →	<b>↓</b> Boundary Conditions → O	perating Conditions
---------	----------------------------------	---------------------

Operating Conditions	<b>—</b>
Pressure	Gravity
Operating Pressure (pascal)	🔲 Gravity
0 P	
Reference Pressure Location	
X (m) 0	
Y (m) 0	
Z (m)	
OK Cancel Hel	>

- a. Enter 0 Pa for **Operating Pressure**.
- b. Click **OK** to close the **Operating Conditions** dialog box.

## 18.4.9. Solution

1. Set the solution parameters.



Solution Methods	
Pressure-Velocity Coupling	
Scheme	
Coupled	
Coupled with Volume Fractions	
Solve N-Phase Volume Fraction Equations	
Spatial Discretization	_
Pressure	^
PRESTO!	
Momentum	
QUICK 🔹	
Volume Fraction	
QUICK 🔹	Ξ
Turbulent Kinetic Energy	
First Order Upwind	
Turbulent Dissipation Rate	
First Order Upwind 🔹	Ŧ
Transient Formulation	
<b></b>	
Non-Iterative Time Advancement	
Frozen Flux Formulation	
🗹 Pseudo Transient	
Warped-Face Gradient Correction	
High Order Term Relaxation Options	
Default	

- a. Select **Coupled** from the **Scheme** drop-down list in the **Pressure-Velocity Coupling** group box.
- b. Retain the selection of **PRESTO!** from the **Pressure** drop-down list in the **Spatial Discretization** group box.
- c. Select **QUICK** for **Momentum** and **Volume Fraction**.
- d. Retain First Order Upwind for Turbulent Kinetic Energy and Turbulent Dissipation Rate.
- e. Enable Pseudo Transient.
- f. Enable High Order Term Relaxation.

The message appears in the console informing you of changing AMG cycle type for Volume Fraction, Turbulent Kinetic Energy, and Turbulent Dissipation Rate to F-cycle.

The relaxation of high order terms will help to improve the solution behavior of flow simulations when higher order spatial discretizations are used (higher than first).

2. Set the solution controls.

# $\begin{tabular}{|c|c|c|c|c|}\hline \hline \begin{tabular}{c|c|c|c|} \hline \begin{tabular}{c|c|c|c|} \hline \begin{tabular}{c|c|c|c|} \hline \begin{tabular}{c|c|c|} \hline \begin{tabular}{c|c|c|} \hline \begin{tabular}{c|c|c|} \hline \begin{tabular}{c|c|} \hline \b$

Solution Controls	
Pseudo Transient Explicit Relaxation Factors	
Pressure	
0.5	L
Momentum	
0.5	
Density	
1	
Body Forces	
1	
Volume Fraction	
0.3	
Default Equations Limits Advanced	-
Help	

- a. Set the pseudo transient explicit relaxation factor for **Volume Fraction** to 0.3.
- 3. Enable the plotting of residuals during the calculation.

### **Solving** $\rightarrow$ **Reports** $\rightarrow$ **Residuals...**

Residual Monitors					×
Options  Print to Console  Plot  Window  1  Curves  Axes  Iterations to Plot  1000	Equations Residual continuity x-velocity y-velocity k	Monitor C	Check Convergence	Absolute Criteria 1e-05 1e-05 1e-05 1e-05	
Iterations to Store	Residual Values           Normalize           Scale           Compute Local	al Scale	Iterations 5	Convergence Cr absolute	iterion
OK Plot Renormalize Cancel Help					

- a. Ensure that **Plot** is enabled in the **Options** group box.
- b. Enter 1e-05 for the Absolute Criteria of continuity, x-velocity, y-velocity, k, and epsilon.

Decreasing the criteria for these residuals will improve the accuracy of the solution.

- c. Click **OK** to close the **Residual Monitors** dialog box.
- 4. Initialize the solution.



- a. Select Hybrid initialization method from the Initialization group.
- b. Click More Settings... to open the Hybrid Initialization dialog box.

Hybrid Initialization	n	<b>—</b>
General Settings	Turbulence Settings	Species Settings
Use External-A	axation Factor	
	OK Cancel Help	

- c. Enable **Use Specified Initial Pressure on Inlets** in the **Initialization Options** group box. The velocity will now be initialized to the **Initial Gauge Pressure** value that you set in the **Pressure Inlet** boundary condition dialog box. For more information on initialization options, see hybrid initialization in the Fluent User's Guide.
- d. Click **OK** to close the **Hybrid Initialization** dialog box.

e. Click **Initialize** to initialize the solution.

#### Note

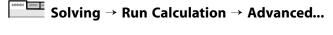
For flows in complex topologies, hybrid initialization will provide better initial velocity and pressure fields than standard initialization. This will help to improve the convergence behavior of the solver.

5. Save the case file (cav.cas.gz).

<b>File</b> $\rightarrow$ Write $\rightarrow$	Case
---	------

- - - ---

6. Start the calculation by requesting 400 iterations.



Run Calculation	
Check Case	Update Dynamic Mesh
Pseudo Transient Opti	ons
Fluid Time Scale	
Time Step Method	Timescale Factor
O User Specified	1
Automatic	
Length Scale Method	Verbosity
Conservative •	0
Number of Iterations	Paparting Internal
Number of Iterations	Reporting Interval
400	Reporting Interval
400 <	
400	
400 Profile Update Interval	
400 Profile Update Interval 1 	1

- a. Enter 400 for **Number of Iterations**.
- b. Click Calculate.

The solution will converge in approximately 340 iterations.

7. Save the data file (cav.dat.gz).

**File**  $\rightarrow$  Write  $\rightarrow$  Data...

### 18.4.10. Postprocessing

1. Create and plot a definition of pressure contours in the orifice (Figure 18.3: Contours of Static Pressure (p. 771)).

### Postprocessing $\rightarrow$ Graphics $\rightarrow$ Contours $\rightarrow$ New...

Contours		×
Contour Name		
contour-static-pressu	re	
Options	Contours of	
V Filled	Pressure	•
Node Values	Static Pressure	•
Global Range	Phase	
Auto Range Clip to Range	mixture	•
Draw Profiles	Min Max	
Draw Mesh	0 0	
Coloring	Surfaces Filter Text	x
<ul> <li>Banded</li> </ul>	default-interior	
Smooth	inlet_1	=
	inlet_2 outlet	
Colormap Options	symm_1	-
	New Surface 🔻	
6	Save/Display Compute Close Help	

- a. Change Contour Name to contour-static-pressure
- b. Enable **Filled** in the **Options** group box.
- c. Retain the default selection of Pressure... and Static Pressure from the Contours of drop-down lists.
- d. Click Save/Display and close the Contours dialog box.

The contour-static-pressure contour definition appears under the **Results/Graphics/Con-tours** tree branch. Once you create a plot definition, you can use a right-click menu to display this definition at a later time, for instance, in subsequent simulations with different settings ;or in combination with other plot definitions.

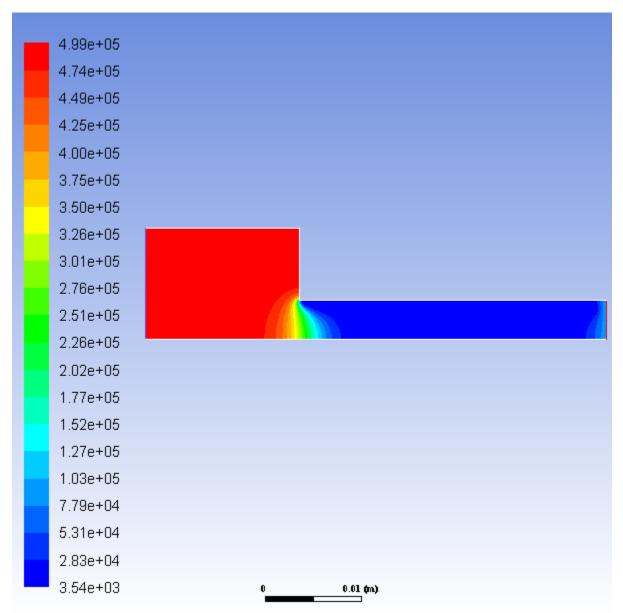


Figure 18.3: Contours of Static Pressure

Note the dramatic pressure drop at the flow restriction in Figure 18.3: Contours of Static Pressure (p. 771). Low static pressure is the major factor causing cavitation. Additionally, turbulence contributes to cavitation due to the effect of pressure fluctuation (Figure 18.4: Mirrored View of Contours of Static Pressure (p. 773)) and turbulent diffusion (Figure 18.5: Contours of Turbulent Kinetic Energy (p. 774)).

2. Mirror the display across the centerline (Figure 18.4: Mirrored View of Contours of Static Pressure (p. 773)).



Mirroring the display across the centerline gives a more realistic view.

<b>U</b> iews				
Views back front Save Name view-0	Actions Default Auto Scale Previous Save Delete Read Write Mirror Planes [2/2] symm_2 symm_1 Define Plane Periodic Repeats Define			
Apply Camera Close Help				

- a. Select symm\_2 and symm\_1 from the Mirror Planes selection list.
- b. Click **Apply** and close the **Views** dialog box.

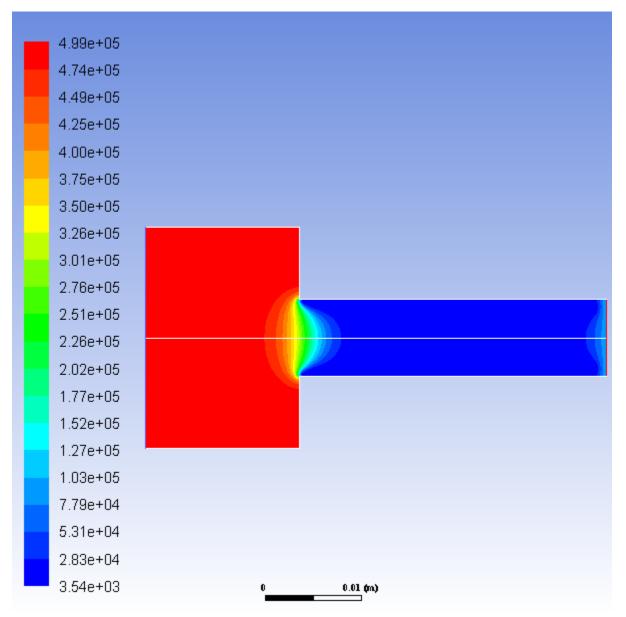
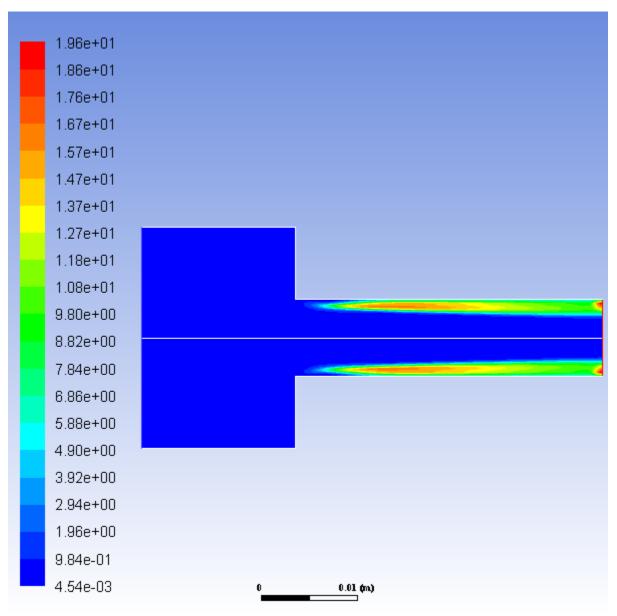


Figure 18.4: Mirrored View of Contours of Static Pressure

3. Create and plot a contour definition of the turbulent kinetic energy (Figure 18.5: Contours of Turbulent Kinetic Energy (p. 774)).

Postprocessing  $\rightarrow$  Graphics  $\rightarrow$  Contours  $\rightarrow$  New...

- a. Change Contour Name to contour-tke
- b. Enable **Filled** in the **Options** group box.
- c. Select Turbulence... and Turbulent Kinetic Energy(k) from the Contours of drop-down lists.
- d. Click Save/Display.



#### Figure 18.5: Contours of Turbulent Kinetic Energy

In this example, the mesh used is fairly coarse. However, in cavitating flows the pressure distribution is the dominant factor, and is not very sensitive to mesh size.

4. Create and plot a contour definition of the volume fraction of water vapor (Figure 18.6: Contours of Vapor Volume Fraction (p. 775)).

Postprocessing  $\rightarrow$  Graphics  $\rightarrow$  Contours  $\rightarrow$  New...

- a. Change Contour Name to contour-vf-vapor
- b. Enable **Filled** in the **Options** group box.
- c. Select Phases... and Volume fraction from the Contours of drop-down lists.
- d. Select vapor from the Phase drop-down list.

e. Click Save/Display and close the Contours dialog box.

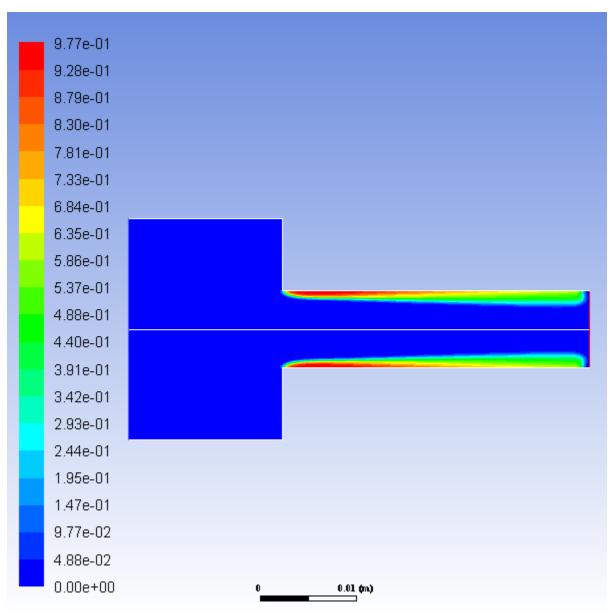


Figure 18.6: Contours of Vapor Volume Fraction

The high turbulent kinetic energy region near the neck of the orifice in Figure 18.5: Contours of Turbulent Kinetic Energy (p. 774) coincides with the highest volume fraction of vapor in Figure 18.6: Contours of Vapor Volume Fraction (p. 775). This indicates the correct prediction of a localized high phase change rate. The vapor then gets convected downstream by the main flow.

### 18.5. Summary

This tutorial demonstrated how to set up and resolve a strongly cavitating pressure-driven flow through an orifice, using multiphase mixture model of ANSYS Fluent with cavitation effects. You learned how to set the boundary conditions for an internal flow. A steady-state solution was calculated to simulate the formation of vapor in the neck of the flow after the section restriction at the orifice. A more computationally intensive transient calculation is necessary to accurately simulate the irregular cyclic process of bubble formation, growth, filling by water jet re-entry, and break-off.

## **18.6. Further Improvements**

This tutorial guides you through the steps to reach an initial solution. You may be able to obtain a more accurate solution by using an appropriate higher-order discretization scheme and by adapting the mesh. Mesh adaption can also ensure that the solution is independent of the mesh. These steps are demonstrated in Introduction to Using ANSYS Fluent: Fluid Flow and Heat Transfer in a Mixing Elbow (p. 121).

# Chapter 19: Using the Mixture and Eulerian Multiphase Models

This tutorial is divided into the following sections:

19.1. Introduction19.2. Prerequisites19.3. Problem Description19.4. Setup and Solution19.5. Summary19.6. Further Improvements

# 19.1. Introduction

This tutorial examines the flow of water and air in a tee junction. Initially you will solve the problem using the less computationally intensive mixture model. You will then switch to the more accurate Eulerian model and compare the results of these two approaches.

This tutorial demonstrates how to do the following:

- · Use the mixture model with slip velocities.
- Set boundary conditions for internal flow.
- Calculate a solution using the pressure-based coupled solver with the mixture model.
- Use the Eulerian model.
- Calculate a solution using the multiphase coupled solver with the Eulerian model.
- Display the results obtained using the two approaches for comparison.

### 19.2. Prerequisites

This tutorial is written with the assumption that you have completed one or more of the introductory tutorials found in this manual:

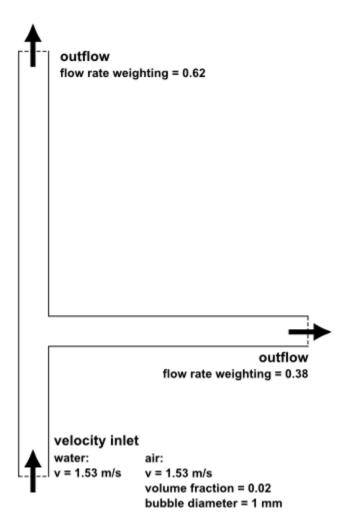
- Introduction to Using ANSYS Fluent in ANSYS Workbench: Fluid Flow and Heat Transfer in a Mixing Elbow (p. 1)
- Parametric Analysis in ANSYS Workbench Using ANSYS Fluent (p. 73)
- Introduction to Using ANSYS Fluent: Fluid Flow and Heat Transfer in a Mixing Elbow (p. 121)

and that you are familiar with the ANSYS Fluent tree and ribbon structure. Some steps in the setup and solution procedure will not be shown explicitly.

# **19.3. Problem Description**

This problem considers an air-water mixture flowing upwards in a duct and then splitting in a tee junction. The ducts are 25 mm in width, the inlet section of the duct is 125 mm long, and the top and the side ducts are 250 mm long. The schematic of the problem is shown in Figure 19.1: Problem Specification (p. 778).

#### Figure 19.1: Problem Specification



# 19.4. Setup and Solution

The following sections describe the setup and solution steps for this tutorial:

19.4.1. Preparation
19.4.2. Mesh
19.4.3. General Settings
19.4.3. General Settings
19.4.4. Models
19.4.5. Materials
19.4.6. Phases
19.4.7. Boundary Conditions
19.4.8. Operating Conditions
19.4.9. Solution Using the Mixture Model
19.4.10. Postprocessing for the Mixture Solution
19.4.11. Higher Order Solution using the Mixture Model

19.4.12. Setup and Solution for the Eulerian Model 19.4.13. Postprocessing for the Eulerian Model

### 19.4.1. Preparation

To prepare for running this tutorial:

- 1. Set up a working folder on the computer you will be using.
- 2. Go to the ANSYS Customer Portal, https://support.ansys.com/training.

#### Note

If you do not have a login, you can request one by clicking **Customer Registration** on the log in page.

- 3. Enter the name of this tutorial into the search bar.
- 4. Narrow the results by using the filter on the left side of the page.
  - a. Click ANSYS Fluent under Product.
  - b. Click 18.0 under Version.
- 5. Select this tutorial from the list.
- 6. Click the **mix\_eulerian\_multiphase\_R180.zip** link to download the input files.
- 7. Unzip mix\_eulerian\_multiphase\_R180.zip to your working folder.

The file tee.msh can be found in the mix\_eulerian\_multiphase folder created after unzipping the file.

8. Use Fluent Launcher to enable **Double Precision** and start the **2D** version of ANSYS Fluent.

Fluent Launcher displays your **Display Options** preferences from the previous session.

For more information about Fluent Launcher, see starting ANSYS Fluent using the Fluent Launcher in the Fluent Getting Started Guide.

- 9. Ensure that the **Display Mesh After Reading** and **Workbench Color Scheme** options are enabled.
- 10. Run in Serial under Processing Options.

#### Note

The double precision solver is recommended for modeling multiphase flow simulations.

#### 19.4.2. Mesh

1. Read the mesh file tee.msh.

**File**  $\rightarrow$  Read  $\rightarrow$  Mesh...

As ANSYS Fluent reads the mesh file, it will report the progress in the console.

### 19.4.3. General Settings

1. Check the mesh.

### **Setting Up Domain** $\rightarrow$ Mesh $\rightarrow$ Check

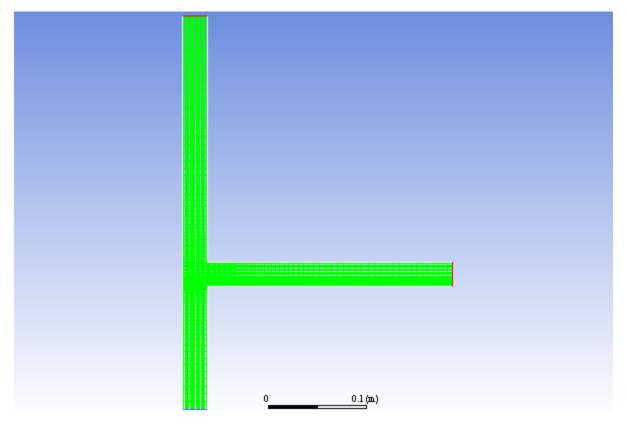
ANSYS Fluent will perform various checks on the mesh and will report the progress in the console. Ensure that the reported minimum volume is a positive number.

2. Examine the mesh (Figure 19.2: Mesh Display (p. 780)).

#### Extra

You can use the right mouse button to probe for mesh information in the graphics window. If you click the right mouse button on any node in the mesh, information will be displayed in the ANSYS Fluent console about the associated zone, including the name of the zone. This feature is especially useful when you have several zones of the same type and you want to distinguish between them quickly.

Figure 19.2: Mesh Display



3. Retain the default settings for the pressure-based solver.

### Setup → General

General	
Mesh	
Scale	Check Report Quality
Display	
Solver	
Туре	Velocity Formulation
Pressure-Based	Absolute
O Density-Based	Relative
Time	2D Space
Steady	Planar
Transient	Axisymmetric
	Axisymmetric Swirl
Gravity Units	
Help	

### 19.4.4. Models

1. Select the mixture multiphase model with slip velocities.

 $\blacksquare Setting Up Physics \rightarrow Models \rightarrow Multiphase...$ 

a. Select Mixture in the Model list.

The **Multiphase Model** dialog box will expand to show the inputs for the mixture model.

🖸 Multiphase Model	<b>×</b>			
Model	Number of Eulerian Phases			
Off Off	2			
Volume of Fluid				
Mixture				
Eulerian				
Wet Steam				
Mixture Parameters				
Slip Velocity				
Body Force Formulation				
Implicit Body Force				
OK Cancel Help				

b. Ensure that Slip Velocity is enabled in the Mixture Parameters group box.

You need to solve the slip velocity equation because there will be significant difference in velocities for the different phases.

c. Enable Implicit Body Force in the Body Force Formulation group box.

This treatment improves solution convergence by accounting for the partial equilibrium of the pressure gradient and body forces in the momentum equations. It is used in VOF and mixture problems, where body forces are large in comparison to viscous and convective forces.

- d. Click OK to close the Multiphase Model dialog box.
- 2. Select the realizable k- $\varepsilon$  turbulence model with standard wall functions.

**Setting Up Physics**  $\rightarrow$  Models  $\rightarrow$  Viscous...

<b>E</b> Viscous Model	<b>—X</b> —				
Model Laminar Spalart-Allmaras (1 eqn) k-epsilon (2 eqn) k-omega (2 eqn) Transition k-kl-omega (3 eqn) Transition SST (4 eqn) Reynolds Stress (5 eqn) Scale-Adaptive Simulation (SAS) Detached Eddy Simulation (DES) k-epsilon Model	Model Constants C2-Epsilon 1.9 TKE Prandtl Number 1 TDR Prandtl Number 1.2 Dispersion Prandtl Number 0.75				
<ul> <li>Standard</li> <li>RNG</li> <li>Realizable</li> </ul>					
Near-Wall Treatment Standard Wall Functions Scalable Wall Functions Non-Equilibrium Wall Functions Enhanced Wall Treatment Menter-Lechner User-Defined Wall Functions					
Options Curvature Correction Production Limiter Mixture Drift Force	User-Defined Functions Turbulent Viscosity None				
OK Cancel Help					

- a. Select **k-epsilon** in the **Model** list.
- b. Select Realizable under in the k-epsilon Model list.

The realizable k- $\varepsilon$  model is recommended in cases where flow separation around sharp corners or over bluff bodies can be expected.

c. Retain Standard Wall Functions in the Near-Wall Treatment list.

This problem does not require a particularly fine mesh, and standard wall functions will be used.

d. Click **OK** to close the **Viscous Model** dialog box.

#### 19.4.5. Materials

1. Copy the properties for liquid water from the materials database so that it can be used for the primary phase.

# **F**Setup $\rightarrow$ Materials $\rightarrow$ Fluid $\rightarrow$ air $\stackrel{\text{D}}{\rightarrow}$ Edit...

a. Click the Fluent Database... button to open the Fluent Database Materials dialog box.

Fluent Database Materials		<b>X</b>
Fluent Fluid Materials [1/563]		Material Type [fluid -
vinyl-silylidene (h2cchsih) vinyl-trichlorosilane (sicl3ch2ch) vinylidene-chloride (ch2ccl2)	•	<ul> <li>Order Materials by</li> <li>Name</li> <li>Chemical Formula</li> </ul>
water-liquid (h2o <l>) water-vapor (h2o) wood-volatiles (wood_vol)</l>		
Copy Materials from Case Delete Properties		,
Density (kg/m3)	constant	View
	998.2	E
Cp (Specific Heat) (j/kg-k)	constant	▼ View
	4182	
Thermal Conductivity (w/m-k)	constant	▼ View
	0.6	
Viscosity (kg/m-s)	·	View
	0.001003	
New Edit	Save Copy Clo	se Help

i. Select water-liquid (h2o < l >) from the Fluent Fluid Materials selection list.

Scroll down the list to find **water-liquid** (h2o < l >).

- ii. Click **Copy** to copy the properties for liquid water to your model.
- iii. Close the Fluent Database Materials dialog box.
- b. Close the Create/Edit Materials dialog box.

#### 19.4.6. Phases

In the following steps you will define the liquid water and air phases that flow in the tee junction.

🖃 🍓 Setup		
General		
🚍 📲 Models		
🚊 📲 Multiphase (Mixture)		
🖃 🔡 Phases		
🔤 🔡 phase-2 - Secondary Ph	Edit	
	Delete	13
📟 🔡 Energy (Off)		
	d Wall Fn)	

1. Specify liquid water as the primary phase.

<b>E</b> Setup $\rightarrow$ Models $\rightarrow$ Multiphase $\rightarrow$ Phases $\rightarrow$ phase-1	<u>ď</u>	Edit
---	----------	------

Primary Phase	×
Name	
water	
Phase Material water-liquid    Edit	
OK Delete Cancel Help	

- a. Enter water for Name.
- b. Select water-liquid from the Phase Material drop-down list.
- c. Click **OK** to close the **Primary Phase** dialog box.
- 2. Specify air as the secondary phase.

**E** Setup  $\rightarrow$  Models  $\rightarrow$  Multiphase  $\rightarrow$  Phases  $\rightarrow$  phase-2  $\stackrel{\square}{\hookrightarrow}$  Edit...

Secondary Phase	×
Name	
air	
Phase Material air 🔹 Edit	
🔲 Granular	
Interfacial Area Concentration	
Properties	
Diameter (m) constant   Edit	
0.001	
OK Delete Cancel Help	
	щ

- a. Enter air for Name.
- b. Retain the default selection of air from the Phase Material drop-down list.
- c. Enter 0.001 m for **Diameter**.
- d. Click OK to close the Secondary Phase dialog box.
- 3. Check that the drag coefficient is set to be calculated using the Schiller-Naumann drag law.

Setup  $\rightarrow$  Models  $\rightarrow$  Multiphase  $\rightarrow$  Phases  $\rightarrow$  Phases Interactions  $\stackrel{\frown}{\rightarrow}$  Edit...

Phase Interaction				×		
Virtual Mass Drag	Lift Wall Lubrication	Turbulent Dispersion	Turbulence Interaction	Collistor		
Drag Coefficient						
air water Edit						
OK Cancel Help						

a. Click the **Drag** tab.

b. Retain the default selection of schiller-naumann from the Drag Coefficient drop-down list.

The Schiller-Naumann drag law describes the drag between the spherical particle and the surrounding liquid for a wide range of conditions provided the bubbles remain approximately spherical. In this case, the bubbles have a diameter of 1 mm which is within the spherical regime.

c. Click **OK** to close the **Phase Interaction** dialog box.

### 19.4.7. Boundary Conditions

Boundary Conditions
Zone Filter Text
default-interior
outflow-3
outflow-5
velocity-inlet-4
wall-1
Phase Type ID
mixture Velocity-inlet Velocity-inlet
Edit Copy Profiles
Parameters Operating Conditions
Display Mesh
Periodic Conditions
Help

For this problem, you need to set the boundary conditions for three boundaries: the velocity inlet and the two outflows. Since this is a mixture multiphase model, you will set the conditions at the velocity inlet that are specific for the mixture (conditions that apply to all phases) and also conditions that are specific to the primary and secondary phases.

1. Set the boundary conditions at the velocity inlet (**velocity-inlet-4**) for the mixture.

Setup  $\rightarrow$  Boundary Conditions  $\rightarrow$  velocity-inlet-4  $\stackrel{\bigcirc}{\rightarrow}$  Edit...

Velocity Inlet								×
Zone Name velocity-inlet-4						Pha mi	ase xture	
Momentum	Thermal	Radiation	Species	DPM	Multipl	nase	Potential	UDS
Supersonic/Initial Gauge Pressure (pascal) 0 constant  Turbulence Specification Method Intensity and Viscosity Ratio Turbulent Intensity (%) 5 Turbulent Viscosity Ratio 10 P OK Cancel Help								

- a. Retain the default selection of Intensity and Viscosity Ratio as the turbulence Specification Method.
- b. Retain the default of 5 % for **Turbulent Intensity**.
- c. Retain the default of 10 for Turbulent Viscosity Ratio.
- d. Click **OK** to close the **Velocity Inlet** dialog box.
- 2. Set the boundary conditions at the velocity inlet (velocity-inlet-4) for the primary phase (water).

Setup $ ightarrow$ Boundary Conditions $ ightarrow$ velocity-inlet-4 $ ightarrow$ water	₫́ Edit
---	---------

💶 Velocity Inlet						×
Zone Name				Pha		_
velocity-inlet-4				wa	ter	
Momentum Thermal	Radiation	Species	DPM	Multiphase	Potential	UDS
Velocity Specification Metho	Velocity Specification Method Magnitude, Normal to Boundary					
Reference Frame Absolute						
Velocity Magnitude (m/s) 1.53 constant						)
OK Cancel Help						

- a. Retain the default selection of **Magnitude**, **Normal to Boundary** from the **Velocity Specification Method** drop-down list.
- b. Retain the default selection of Absolute from the Reference Frame drop-down list.
- c. Enter 1.53 m/s for **Velocity Magnitude**.

d. Click **OK** to close the **Velocity Inlet** dialog box.

#### Note

Note that you can also open the **Velocity Inlet** dialog box by double-clicking the **Setup/Boundary Conditions/velocity-inlet-4/water** tree item.

3. Set the boundary conditions at the velocity inlet (velocity-inlet-4) for the secondary phase (air).

**E** Setup  $\rightarrow$  Boundary Conditions  $\rightarrow$  velocity-inlet-4  $\rightarrow$  air  $\stackrel{0}{\xrightarrow{}}$  Edit...

- a. Retain the default selection of **Magnitude**, **Normal to Boundary** from the **Velocity Specification Method** drop-down list.
- b. Retain the default selection of Absolute from the Reference Frame drop-down list.
- c. Enter 1.53 m/s for **Velocity Magnitude**.

In multiphase flows, the volume rate of each phase is usually known. Volume rate divided by the inlet area gives the superficial velocity, which is the product of the inlet physical velocity and the volume fraction. When you have two phases, you must enter two physical velocities and the volume fraction of the secondary phase. Here it is assumed that bubbles at the inlet are moving at the same physical speed as the water.

d. Click the **Multiphase** tab and enter 0.02 for **Volume Fraction**.

💶 Velocity Inlet							×
Zone Name						ase	_
velocity-inlet-4				-	air	- -	
Momentum	Thermal	Radiation	Species	DPM	Multiphase	Potential	UDS
Volume Fraction	0.02		consta	nt	•		
		OK	Cancel	Help			

e. Click OK to close the Velocity Inlet dialog box.

#### Note

Note that you can also open the **Velocity Inlet** dialog box by double-clicking the **Setup/Boundary Conditions/velocity-inlet-4/air** tree item.

4. Set the boundary conditions at **outflow-5** for the mixture.

**Setup**  $\rightarrow$  **Boundary Conditions**  $\rightarrow$  **outflow-5**  $\stackrel{\textcircled{0}}{\rightarrow}$  **Edit...** 

- a. Click Edit... to open the Outflow dialog box.
  - i. Enter 0.62 for Flow Rate Weighting.
  - ii. Click **OK** to close the **Outflow** dialog box.
- 5. Set the boundary conditions at **outflow-3** for the mixture.



- a. Enter 0.38 for Flow Rate Weighting.
- b. Click **OK** to close the **Outflow** dialog box.

### 19.4.8. Operating Conditions

1. Set the gravitational acceleration.

٠

Setup → <sup>•</sup>	🖵 Boundary	Conditions $\rightarrow$	Operating	Conditions
----------------------	------------	--------------------------	-----------	------------

Operating Conditions	<b>—</b>
Pressure Operating Pressure (pascal) 101325 P Reference Pressure Location X (m) 0 P Y (m) 0 P Z (m) 0 P	Gravity ♥ Gravity Gravitational Acceleration X (m/s2) 0 Y (m/s2) -9.81 Z (m/s2) 0 Variable-Density Parameters ♥ Specified Operating Density Operating Density (kg/m3) 0 ₽
ОК Саг	ncel Help

a. Enable **Gravity**.

The **Operating Conditions** dialog box will expand to show additional inputs.

- b. Enter  $-9.81 \text{ m/s}^2$  for **Y** in the **Gravitational Acceleration** group box.
- c. Enable Specified Operating Density.
- d. Enter 0 kg/m<sup>3</sup> for **Operating Density**.

ANSYS Fluent redefines the fluid pressure by removing the hydrostatic component based on an average density in the domain or a user-specified operating density. By setting the operating density to 0 you force the hydrostatic pressure to appear explicitly in the postprocessed results. For more information, refer to the Fluent User's Guide.

e. Click **OK** to close the **Operating Conditions** dialog box.

#### **19.4.9. Solution Using the Mixture Model**

You will begin by calculating a preliminary solution using first-order discretization for momentum, volume fraction and turbulence quantities. You will then change to higher-order methods to refine the solution.

1. Set the solution parameters.

<b>Solving</b> $\rightarrow$ Solution $\rightarrow$ Methods
---

Solution Methods	
Pressure-Velocity Coupling	
Scheme	
Coupled	•
Solve N-Phase Volume Fraction Equations	
Spatial Discretization	
Gradient	^
Least Squares Cell Based 🔹	
Pressure	
PRESTO!	
Momentum	Ξ
First Order Upwind 🔹	
Volume Fraction	
First Order Upwind 🔹	
Turbulent Kinetic Energy	
First Order Upwind 🔹	
Turkulant Discinction Data	-
Transient Formulation	
Non-Iterative Time Advancement	
Frozen Flux Formulation	
Pseudo Transient	
Warped-Face Gradient Correction	
High Order Term Relaxation Options	
Default	

- a. Select **Coupled** from the **Scheme** drop-down list.
- b. Confirm that **PRESTO!** is selected from the **Pressure** drop-down list.

Solving  $\rightarrow$  Solution  $\rightarrow$  Controls...

The PRESTO! method for pressure is a good choice when buoyancy and inertial forces are present.

2. Set the solution controls.

100001 10000 10

Solutio	on Controls				
Flow Co	ourant Numbe	er			
40					
	-Explicit Rela	axation	Factors		
	Momentum	0.5			
	Pressure	0.5			
Under-I	Relaxation Fac	tors			
Den	sity				
1					
Body	y Forces				
1					=
Slip	Velocity				
0.1					
Volu	me Fraction				
0.4					
Turb	ulent Kinetic	Energ	y		
0.8					
		n-	**		Ŧ
Defau	lt				
Equat	ions Limit	ts	Advance	ed	

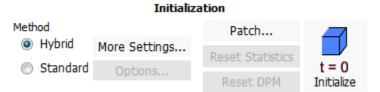
- a. Enter 40 for Flow Courant Number.
- b. Enter 0.4 for Volume Fraction in the Under-Relaxation Factors group box.
- 3. Enable the plotting of residuals during the calculation.

10001 10001 10	Solving $\rightarrow$	Reports →	Residuals
----------------	-----------------------	-----------	-----------

Residual Monitors					×
Options	Equations				
Print to Console	Residual	Monito	r Check Converge	nce Absolute Criteria	
V Plot	continuity	<b>V</b>	$\checkmark$	1e-05	
Window	x-velocity	<b>V</b>	$\checkmark$	0.001	E
1 🗘 Curves Axes	y-velocity	<b>V</b>		0.001	
Iterations to Plot	k	<b>V</b>		0.001	
1000	4		[TT#]	0.004	
1000	Residual Values			Convergence Criterior	1
	Normalize		Iterations	absolute	•
Iterations to Store			5		
1000 🜩	Scale			Convergence Condit	ions
	Compute Loca	l Scale			
ОК	Plot Renormaliz	e Car	ncel Help		

- a. Ensure that **Plot** is enabled in the **Options** group box.
- b. Enter 1e-05 for Absolute Criteria for continuity.
- c. Click OK to close the Residual Monitors dialog box.
- 4. Initialize the solution.

```
Solving \rightarrow Initialization \rightarrow Hybrid
```



- a. Select Hybrid as the initialization Method (Initialization group).
- b. Click Initialize.

#### Note

For flows in complex topologies, hybrid initialization will provide better initial velocity and pressure fields than standard initialization. In general, this will help in improving the convergence behavior of the solver.

5. Save the case file (tee\_la.cas.gz).

```
File \rightarrow Write \rightarrow Case...
```

6. Start the calculation by requesting 1400 iterations.

Solving → Run Calculation

7. Save the case and data files (tee\_la.cas.gz and tee\_la.dat.gz).

File → Write → Case & Data...

8. Check the total mass flow rate for each phase.

Postprocessing  $\rightarrow$  Reports  $\rightarrow$  Fluxes...

E Flux Reports			×
Options Mass Flow Rate	Boundaries Filter Text	Results	
Total Heat Transfer Rate	default-interior		
Radiation Heat Transfer Rate	outflow-3 outflow-5	-14.06729363734643 -23.35083670680045	
	velocity-inlet-4	37.41846187989216	
	wall-1		
Phase mixture			-
		Net Results (kg/s)	
Save Output Parameter		0.0003315357	
C	ompute Write Close Help		

- a. Retain the default selection of Mass Flow Rate in the Options list.
- b. Select water from the Phase drop-down list.
- c. Select outflow-3, outflow-5, and velocity-inlet-4 from the Boundaries selection list.
- d. Click Compute.

Note that the net mass flow rate of water is a small fraction of the inlet and outlet flow rates (<0.1%), indicating that mass is conserved.

e. Select air from the Phase drop-down list and click Compute again.

Again, note that the net mass flow rate of air is small compared to the inlet and outlet flow rates.

f. Close the **Flux Reports** dialog box.

### **19.4.10.** Postprocessing for the Mixture Solution

1. Display the static pressure field in the tee (Figure 19.3: Contours of Static Pressure (p. 796)).

Contours	
Options	Contours of
Filled	Pressure
Node Values	Static Pressure 👻
<ul> <li>Global Range</li> <li>Auto Range</li> </ul>	Phase
Clip to Range	mixture 🔹
Draw Profiles	Min (pascal) Max (pascal)
Draw Mesh	-3273.409 42.48868
Coloring	Surfaces Filter Text
<ul> <li>Banded</li> </ul>	default-interior
Smooth	outflow-3
Levels Setup	velocity-inlet-4
20 💠 1 🚔	wall-1
	New Surface
	Display Compute Close Help

- a. Enable **Filled** in the **Options** group box.
- b. Retain the default selection of **Pressure...** and **Static Pressure** from the **Contours of** drop-down lists.
- c. Click **Display**.

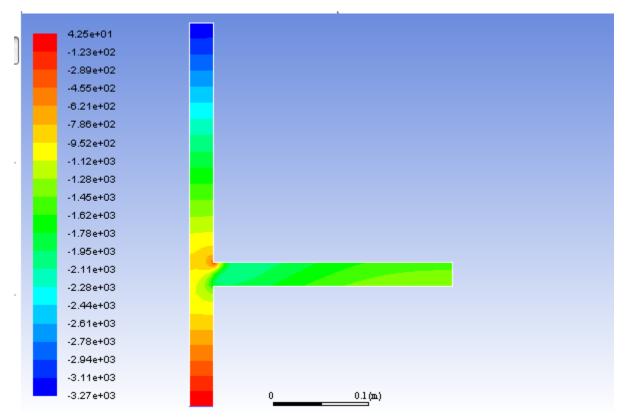


Figure 19.3: Contours of Static Pressure

In Figure 19.3: Contours of Static Pressure (p. 796) the hydrostatic pressure gradient is readily apparent in the vertical arm — a result of setting the Operating Density to 0.

2. Display contours of velocity magnitude (Figure 19.4: Contours of Velocity Magnitude (p. 797)).

- a. Select Velocity... and Velocity Magnitude from the Contours of drop-down lists.
- b. Select water from the Phase drop-down list.
- c. Click Display.

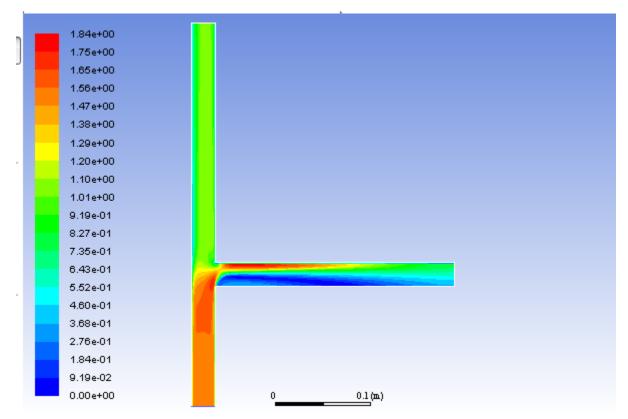


Figure 19.4: Contours of Velocity Magnitude

3. Display the volume fraction of air (Figure 19.5: Contours of Air Volume Fraction (p. 798)).

- a. Select Phases... and Volume fraction from the Contours of drop-down lists.
- b. Select air from the Phase drop-down list.
- c. Click **Display** and close the **Contours** dialog box.

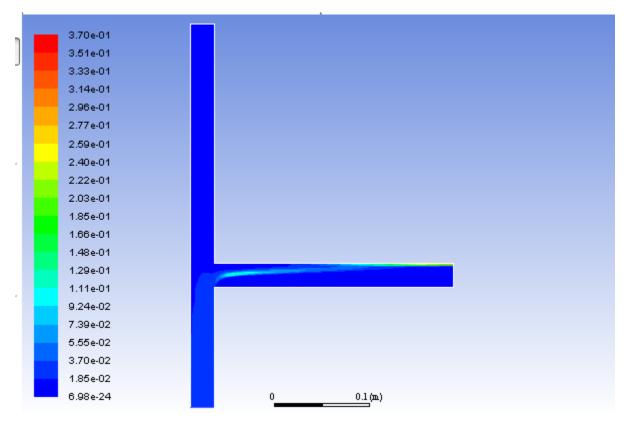


Figure 19.5: Contours of Air Volume Fraction

When gravity acts downwards, it induces stratification in the side arm of the tee junction. In Figure 19.5: Contours of Air Volume Fraction (p. 798), you can see that the gas (air) tends to concentrate on the upper part of the side arm. In this case, gravity acts against inertia that tends to concentrate gas on the low pressure side, thereby creating gas pockets. In the vertical arm, both the gas and the water have velocities in the same direction, and therefore there is no separation. The outflow split modifies the relation between inertia forces and gravity to a large extent, and has an important role in flow distribution and on the gas concentration.

### 19.4.11. Higher Order Solution using the Mixture Model

In this step you will change to higher order discretization schemes and continue the calculation to refine the solution.

1. Revisit the Solution Methods task page and make the following selections:

Group	Setting	Value
Spatial Discretization	Pressure	PRESTO!
	Momentum	Third-Order MUSCL
	Volume Fraction	QUICK
	Turbulent Kinetic Energy	Third-Order MUSCL
	<b>Turbulent Dissipation Rate</b>	Third-Order MUSCL

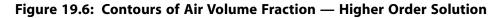
- 2. Run the calculation for an additional 1400 iterations.
- 3. Save the case and data files as tee\_1b.cas.gz and tee\_1b.dat.gz

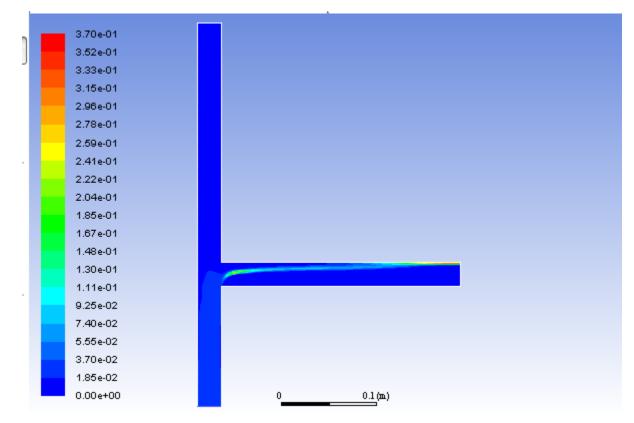
#### **File** $\rightarrow$ Write $\rightarrow$ Case & Data...

4. Plot the contours of air volume fraction using the higher order method on the same scale as in Figure 19.5: Contours of Air Volume Fraction (p. 798).

Postprocessing  $\rightarrow$  Graphics  $\rightarrow$  Contours  $\rightarrow$  Edit...

- a. Select Phases... and Volume fraction from the Contours of drop-down lists.
- b. Select air from the Phase drop-down list.
- c. Disable Auto Range and Clip to Range.
- d. Enter 0 and 3.70e-1 for Min and Max, respectively.
- e. Click **Display** and close the **Contours** dialog box.





### 19.4.12. Setup and Solution for the Eulerian Model

The mixture model is a simplification of the Eulerian model and is valid only when bubble inertia can be neglected. This assumption can be violated in the recirculation pattern. The Eulerian model also offers models for various non-drag forces that are not available when using the mixture model. As a result, the Eulerian model is expected to make a more realistic prediction in this case. You will use the solution obtained using the mixture model as an initial condition for the calculation using the Eulerian model. Because you have already computed a reasonable initial solution, you will continue with the higher order discretization methods.

1. Select the Eulerian multiphase model.

**Setting Up Physics**  $\rightarrow$  Models  $\rightarrow$  Multiphase...

Multiphase Model			<b>—</b>
Model Off Volume of Fluid Mixture Eulerian Vet Steam		Number of Eulerian P	hases T
Eulerian Parameters Dense Discrete Pha Boiling Model Evaporation-Conde Multi-Fluid VOF Mo	ensation		
Volume Fraction Para Formulation Explicit Implicit	meters		
OK Cancel Help			

- a. Select Eulerian in the Model list.
- b. Click **OK** to close the **Multiphase Model** dialog box.
- 2. Specify the drag and lift laws to be used for computing the interphase momentum transfer.

E Setup $ ightarrow$ Models $ ightarrow$ Multiphase $ ightarrow$ Phases $ ightarrow$ Phases Interactions $\stackrel{\bigcup}{ ightarrow}$ Edition
---

a. In the **Drag** tab, retain the default selection of **schiller-naumann** from the **Drag Coefficient** dropdown list.

17

b. In the Lift tab, select legendre-magnaudet from the Lift Coefficient drop-down list.

Lift forces can arise when the gradient of the primary phase velocity field has a component normal to the bubble flow.

c. Click OK to close the Phase Interaction dialog box.

#### Note

For this problem, there are no parameters to be set for the individual phases other than those that you specified when you set up the phases for the mixture model calculation. If you use the Eulerian model for a flow involving a granular secondary phase, you will need to set additional parameters. There are also other options in the **Phase Interaction** dialog box that may be relevant for other applications.

For details on setting up an Eulerian multiphase calculation, see the Fluent User's Guide.

3. Select the multiphase turbulence model.

<b>U</b> iscous Model	×			
Model	Model Constants C2-Epsilon			
<ul> <li>k-epsilon (2 eqn)</li> <li>k-omega (2 eqn)</li> <li>Reynolds Stress (5 eqn)</li> <li>k-epsilon Model</li> </ul>	1.9 TKE Prandtl Number			
	1 TDR Prandtl Number			
<ul> <li>Standard</li> <li>RNG</li> <li>Realizable</li> </ul>	1.2 Dispersion Prandtl Number 0.75			
<ul> <li>Near-Wall Treatment</li> <li>Standard Wall Functions</li> <li>Scalable Wall Functions</li> <li>Non-Equilibrium Wall Functions</li> <li>Enhanced Wall Treatment</li> <li>Menter-Lechner</li> <li>User-Defined Wall Functions</li> <li>Options</li> <li>Curvature Correction</li> <li>Production Limiter</li> </ul>	0.75			
	User-Defined Functions Turbulent Viscosity mixture none			
	water none -			
<ul> <li>Turbulence Multiphase Model</li> <li>Mixture</li> <li>Dispersed</li> <li>Per Phase</li> </ul>				
<u>ا</u>				
OK Cancel Help				

 $\blacksquare Setting Up Physics \rightarrow Models \rightarrow Viscous...$ 

a. Retain the default selection of Mixture in the Turbulence Multiphase Model list.

In this case the dispersed phase volume concentration is relatively small so the mixture turbulence model is sufficient to capture the important features of the turbulent flow.

- b. Click **OK** to close the **Viscous Model** dialog box.
- 4. Confirm that the solution parameters are set to use the higher-order discretization schemes.

Revisit the **Solution Methods** task page and verify that the settings are as follows:

Group	Setting	Value
Pressure-Velocity Coupling	Scheme	Coupled
Spatial Discretization	Momentum	Third-Order MUSCL
	Volume Fraction	QUICK
	Turbulent Kinetic Energy	Third-Order MUSCL
	<b>Turbulent Dissipation Rate</b>	Third-Order MUSCL

#### 5. Set the solution controls

$\diamondsuit$ Solution Cont	trols	
Solution Controls	5	
Flow Courant Numb	per	
40		
Explicit Re	laxation Factors	
Momentum	0.5	
Pressure	0.5	
Under-Relaxation Fa	actors	
Density		*
1		
Body Forces		
1		
Volume Fraction		
0.4		
Turbulent Kinetic Energy		
0.8		
Turbulent Dissipation Rate		
0.8		
Turkulant Minan	a.	Ŧ
Default		
Equations	its Advanced	

- a. Enter 40 for Flow Courant Number.
- b. Enter 0.5 for Momentum and for Pressure in the Explicit Relaxation Factors group box.
- c. Confirm that Volume Fraction is set to 0.4 in the Under-Relaxation Factors group box.
- 6. Continue the solution by requesting 1400 additional iterations.

#### **Solving** $\rightarrow$ Run Calculation $\rightarrow$ Calculate

7. Check that the mass imbalance is small (less than about 0.2 %) using the **Flux Reports** dialog box as for the Mixture model solution.

Postprocessing  $\rightarrow$  Reports  $\rightarrow$  Fluxes...

8. Save the case and data files (tee\_2.cas.gz and tee\_2.dat.gz).

**File**  $\rightarrow$  Write  $\rightarrow$  Case & Data...

#### 19.4.13. Postprocessing for the Eulerian Model

1. Display the static pressure field in the tee for the mixture (Figure 19.7: Contours of Static Pressure — Eulerian Model (p. 804)).

Postprocessing  $\rightarrow$  Graphics  $\rightarrow$  Contours  $\rightarrow$  Edit...

Contours		<b>×</b>	
Options	Contours of		
Filled	Pressure		
Node Values	Static Pressure 👻		
Global Range Auto Range	Phase		
Clip to Range	mixture 🔹		
Draw Profiles	Min	Max	
Draw Mesh	-3.27e3	4.23e1	
Coloring	Surfaces Filter Text		
<ul> <li>Banded</li> <li>Smooth</li> </ul>	default-interior outflow-3 outflow-5		
Levels Setup 20 🖈 1 🜩	velocity-inlet-4 wall-1		
New Surface 🔻			
Display Compute Close Help			

a. Select **Pressure...** from the **Contours of** drop-down list.

By default, **Dynamic Pressure** will be displayed in the lower **Contours of** drop-down list. This will automatically change to **Static Pressure** after you select the appropriate phase in the next step.

b. Select mixture from the Phase drop-down list.

The lower **Contours of** drop-down list will now display **Static Pressure**.

c. As before, disable **Auto Range** (**Clip to Range** will be enabled) and set the **Min** and **Max** values to match those in Figure 19.3: Contours of Static Pressure (p. 796).

#### d. Click **Display**.

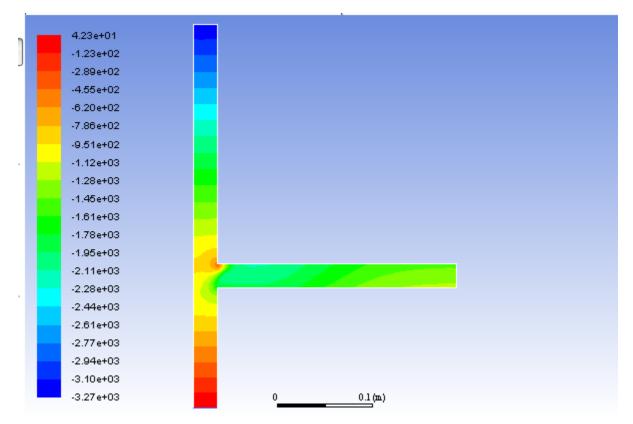


Figure 19.7: Contours of Static Pressure — Eulerian Model

2. Display contours of velocity magnitude for water (Figure 19.8: Contours of Water Velocity Magnitude — Eulerian Model (p. 805)).

Postprocessing  $\rightarrow$  Graphics  $\rightarrow$  Contours  $\rightarrow$  Edit...

- a. Select Velocity... and Velocity Magnitude from the Contours of drop-down lists.
- b. Retain the selection of water from the Phase drop-down list.

Since the Eulerian model solves individual momentum equations for each phase, you can choose the phase for which solution data is plotted.

- c. Set the range to match that in Figure 19.4: Contours of Velocity Magnitude (p. 797).
- d. Click Display.

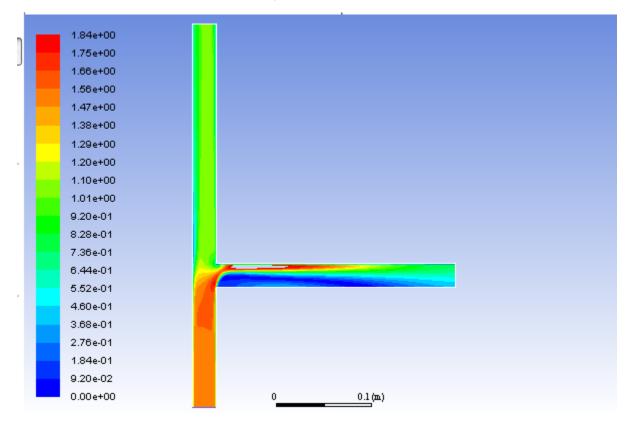
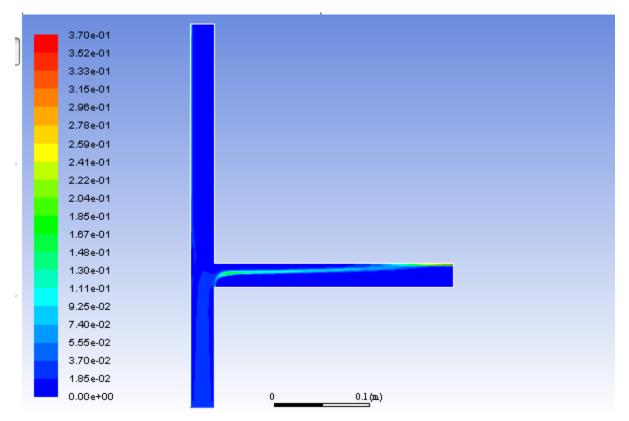


Figure 19.8: Contours of Water Velocity Magnitude — Eulerian Model

3. Display the volume fraction of air (Figure 19.9: Contours of Air Volume Fraction — Eulerian model (p. 806)).

- a. Select Phases... and Volume fraction from the Contours of drop-down lists.
- b. Select air from the Phase drop-down list.
- c. Set the range to match that in Figure 19.5: Contours of Air Volume Fraction (p. 798).
- d. Click **Display** and close the **Contours** dialog box.



#### Figure 19.9: Contours of Air Volume Fraction — Eulerian model

Compare the volume fraction plot in Figure 19.9: Contours of Air Volume Fraction — Eulerian model (p. 806) with the volume fraction plot using the mixture model in Figure 19.6: Contours of Air Volume Fraction — Higher Order Solution (p. 799). Notice that the path of the concentrated air stream in the side arm extends farther into the side arm before drifting to the top surface. As is apparent from the velocity plots, there is a substantial velocity gradient across the side arm as a result of the recirculation near the lower corner of the tee junction. As the dispersed phase bubbles travel along the side arm with the flow, this velocity gradient induces a lift force which tends to oppose the buoyancy force thereby delaying the accumulation of the air concentration along the top surface of the side arm.

### 19.5. Summary

This tutorial demonstrated how to set up and solve a multiphase problem using the mixture model and the Eulerian model. You learned how to set boundary conditions for the mixture and both phases. The solution obtained with the mixture model was used as a starting point for the calculation with the Eulerian model. After completing calculations for each model, you displayed the results to allow for a comparison of the two approaches. For more information about the mixture and Eulerian models, see the Fluent User's Guide.

### **19.6. Further Improvements**

This tutorial guides you through the steps to reach an initial set of solutions. You may be able to obtain a more accurate solution by adapting the mesh. Mesh adaption can also ensure that the solution is independent of the mesh. These steps are demonstrated in Introduction to Using ANSYS Fluent: Fluid Flow and Heat Transfer in a Mixing Elbow (p. 121).

# **Chapter 20: Modeling Solidification**

This tutorial is divided into the following sections: 20.1. Introduction 20.2. Prerequisites 20.3. Problem Description 20.4. Setup and Solution 20.5. Summary 20.6. Further Improvements

# 20.1. Introduction

This tutorial illustrates how to set up and solve a problem involving solidification and will demonstrate how to do the following:

- Define a solidification problem.
- · Define pull velocities for simulation of continuous casting.
- Define a surface tension gradient for Marangoni convection.
- Solve a solidification problem.

# 20.2. Prerequisites

This tutorial is written with the assumption that you have completed one or more of the introductory tutorials found in this manual:

- Introduction to Using ANSYS Fluent in ANSYS Workbench: Fluid Flow and Heat Transfer in a Mixing Elbow (p. 1)
- Parametric Analysis in ANSYS Workbench Using ANSYS Fluent (p. 73)
- Introduction to Using ANSYS Fluent: Fluid Flow and Heat Transfer in a Mixing Elbow (p. 121)

and that you are familiar with the ANSYS Fluent tree and ribbon structure. Some steps in the setup and solution procedure will not be shown explicitly.

# 20.3. Problem Description

This tutorial demonstrates the setup and solution procedure for a fluid flow and heat transfer problem involving solidification, namely the Czochralski growth process. The geometry considered is a 2D axisymmetric bowl (shown in Figure 20.1: Solidification in Czochralski Model (p. 808)), containing liquid metal. The bottom and sides of the bowl are heated above the liquidus temperature, as is the free surface of the liquid. The liquid is solidified by heat loss from the crystal and the solid is pulled out of the domain at a rate of 0.001 m/s and a temperature of 500 K. There is a steady injection of liquid at

the bottom of the bowl with a velocity of  $1.01 \times 10^{-3} m/s$  and a temperature of 1300 K. Material properties are listed in Figure 20.1: Solidification in Czochralski Model (p. 808).

Starting with an existing 2D mesh, the details regarding the setup and solution procedure for the solidification problem are presented. The steady conduction solution for this problem is computed as an initial condition. Then, the fluid flow is enabled to investigate the effect of natural and Marangoni convection in a transient fashion.

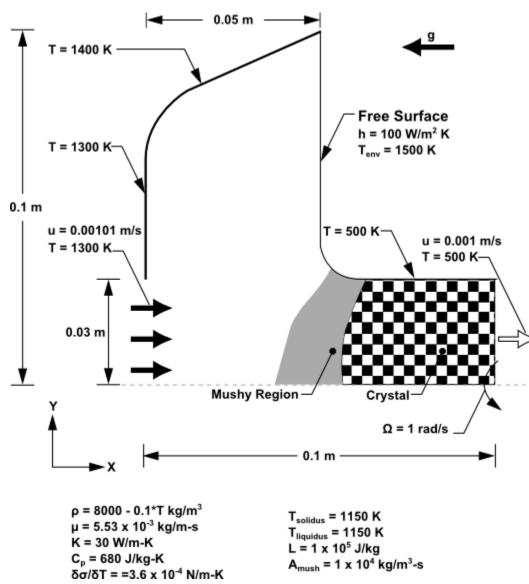


Figure 20.1: Solidification in Czochralski Model

In the above figure,  $A_{mush}$  is the mushy zone constant. For details on modeling the solidification/melting process, refer to momentum equations in the Fluent Theory Guide.

# 20.4. Setup and Solution

The following sections describe the setup and solution steps for this tutorial:

20.4.1. Preparation20.4.2. Reading and Checking the Mesh20.4.3. Specifying Solver and Analysis Type

20.4.4. Specifying the Models
20.4.5. Defining Materials
20.4.6. Setting the Cell Zone Conditions
20.4.7. Setting the Boundary Conditions
20.4.8. Solution: Steady Conduction
20.4.9. Solution: Transient Flow and Heat Transfer

### 20.4.1. Preparation

To prepare for running this tutorial:

- 1. Set up a working folder on the computer you will be using.
- 2. Go to the ANSYS Customer Portal, https://support.ansys.com/training.

#### Note

If you do not have a login, you can request one by clicking **Customer Registration** on the log in page.

- 3. Enter the name of this tutorial into the search bar.
- 4. Narrow the results by using the filter on the left side of the page.
  - a. Click ANSYS Fluent under Product.
  - b. Click **18.0** under **Version**.
- 5. Select this tutorial from the list.
- 6. Click the **solidification\_R180.zip** link to download the input files.
- 7. Unzip the solidification\_R180.zip file you downloaded to your working folder.

The file solid.msh can be found in the solidification directory created after unzipping the file.

8. Use Fluent Launcher to start the **2D** single precision (disable **Double Precision**) version of ANSYS Fluent.

Fluent Launcher displays your **Display Options** preferences from the previous session.

For more information about the Fluent Launcher, see starting ANSYS Fluent using the Fluent Launcher in the Fluent Getting Started Guide.

- 9. Ensure that the **Display Mesh After Reading** and **Workbench Color Scheme** options are enabled.
- 10. Ensure that the Serial processing option is selected.

### 20.4.2. Reading and Checking the Mesh

1. Read the mesh file solid.msh.

**File**  $\rightarrow$  Read  $\rightarrow$  Mesh...

As the mesh is read by ANSYS Fluent, messages will appear in the console reporting the progress of the reading.

A warning about the use of axis boundary conditions is displayed in the console. You are asked to consider making changes to the zone type or change the problem definition to axisymmetric. You will change the problem to axisymmetric swirl later in this tutorial.

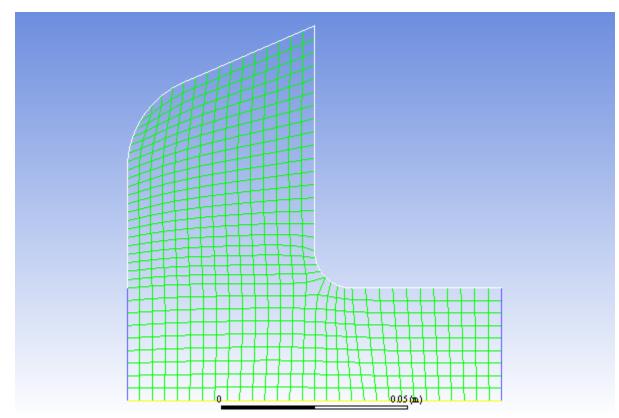
2. Check the mesh.

#### **Setting Up Domain** $\rightarrow$ Mesh $\rightarrow$ Check

ANSYS Fluent will perform various checks on the mesh and will report the progress in the console. Make sure that the minimum volume is a positive number.

3. Examine the mesh (Figure 20.2: Mesh Display (p. 810)).

Figure 20.2: Mesh Display



# 20.4.3. Specifying Solver and Analysis Type

1. Select Axisymmetric Swirl from the 2D Space list.

E Setup → ♀General

General	
Mesh	
Scale	Check Report Quality
Display	
Solver	
Туре	Velocity Formulation
Pressure-Based	Absolute
Density-Based	Relative
Time	2D Space
Steady	Planar
Transient	Axisymmetric
	Axisymmetric Swirl
Gravity Units	
Help	

The geometry comprises an axisymmetric bowl. Furthermore, swirling flows are considered in this problem, so the selection of **Axisymmetric Swirl** best defines this geometry.

Also, note that the rotation axis is the X axis. Hence, the X direction is the axial direction and the Y direction is the radial direction. When modeling axisymmetric swirl, the swirl direction is the tangential direction.

2. Add the effect of gravity on the model.

**F**Setup  $\rightarrow \clubsuit$  General  $\rightarrow \boxed{}$  Gravity

General		
Mesh		
Scale	Check	Report Quality
Display		
Solver		
Туре	Veloci	ty Formulation
Pressure-Ba	sed 💿 A	bsolute
Density-Bas	ed 🔘 R	elative
Time	2D Sp	ace
Steady	P	anar
Transient		xisymmetric
	A	xisymmetric Swirl
Gravity	Units	
Gravitational Ac	celeration	
X (m/s2) -9.81		Ρ
Y (m/s2) 0		P
Z (m/s2) 0		P
Help		
Help		

- a. Enable Gravity.
- b. Enter  $-9.81 \text{ } m/s^2$  for **X** in the **Gravitational Acceleration** group box.

**F** Setup  $\rightarrow$  Models  $\rightarrow$  Solidification & Melting  $\stackrel{\frown}{\hookrightarrow}$  Edit...

# 20.4.4. Specifying the Models

1. Define the solidification model.

Solidification and Melting							
Model	Parameters						
Solidification/Melting	Mushy Zone Parameter constant   Edit						
Back Diffusion	100000						
	Include Pull Velocities						
	Compute Pull Velocities						
OK Cancel Help							

a. Enable the Solidification/Melting option in the Solidification and Melting dialog box.

The Solidification and Melting dialog box will expand to show the related parameters.

b. Retain the default value of 100000 for the Mushy Zone Constant.

This default value is acceptable for most cases.

c. Enable the Include Pull Velocities option.

By including the pull velocities, you will account for the movement of the solidified material as it is continuously withdrawn from the domain in the continuous casting process.

When you enable this option, the **Solidification and Melting** dialog box will expand to show the **Compute Pull Velocities** option. If you were to enable this additional option, ANSYS Fluent would compute the pull velocities during the calculation. This approach is computationally expensive and is recommended only if the pull velocities are strongly dependent on the location of the liquid-solid interface. In this tutorial, you will patch values for the pull velocities instead of having ANSYS Fluent compute them.

For more information about computing the pull velocities, see the Fluent User's Guide.

d. Click OK to close the Solidification and Melting dialog box.

An **Information** dialog box opens, telling you that available material properties have changed for the solidification model. You will set the material properties later, so you can click **OK** in the dialog box to acknowledge this information.

#### Note

ANSYS Fluent will automatically enable the energy calculation when you enable the solidification model, so you need not visit the **Energy** dialog box.

## 20.4.5. Defining Materials

In this step, you will create a new material and specify its properties, including the melting heat, solidus temperature, and liquidus temperature.

1. Define a new material.



Create/Edit Materials		<b></b>
Name	Material Type	Order Materials by
liquid-metal	(fluid 💌	Name
Chemical Formula	Fluent Fluid Materials	Chemical Formula
	lquid-metal 🔹	Fluent Database
	Mixture	
	none 👻	User-Defined Database
Properties		
Density (kg/m3) polyn	omial   Edit	E
Cp (Specific Heat) (j/kg-k) const	ant 💌 Edit 🗧	
1006	43	
Thermal Conductivity (w/m-k) const	ant 🔻 Edit	
0.024	2	
Viscosity (kg/m-s) const	ant 🔻 Edit	
1.789	4e-05	-
	Change/Create Delete Close Heb	

- a. Enter liquid-metal for Name.
- b. Select **polynomial** from the **Density** drop-down list in the **Properties** group box.
- c. Configure the following settings In the **Polynomial Profile** dialog box:

Polynomial Prof	ile		<b>.</b>
Define		In Terms of	Coefficients
Density		Temperature	▼ 2 🚔
Coefficients			
1 8000	2 -0.1	3	4
5	6	7	8
		OK Cancel Help	

- i. Set **Coefficients** to 2.
- ii. In the **Coefficients** group box, enter 8000 for **1** and -0.1 for **2**.

As shown in Figure 20.1: Solidification in Czochralski Model (p. 808), the density of the material is defined by a polynomial function:  $\rho = 8000 - 0.1T$ .

- iii. Click **OK** to close the **Polynomial Profile** dialog box.
- d. In the **Question** dialog box, click **Yes** to overwrite **fluid-1** and add the new material (**liquid-metal**) to the **Fluent Fluid Materials** drop-down list.
- e. Click **OK** in the information dialog box.
- f. Select liquid-metal from the Fluent Fluid Materials drop-down list to set the other material properties.

Create/Edit Materials		<b>—</b>
Name	Material Type	Order Materials by
liquid-metal	1 comments	<ul> <li>Name</li> <li>Chemical Formula</li> </ul>
Chemical Formula	Fluent Fluid Materials	Chemical Formula
	liquid-metal	Fluent Database
	Mixture	
	none	User-Defined Database
Properties		
Viscosity (kg/m-s)	constant	
	0.00553	
Pure Solvent Melting Heat (j/kg)	constant	
	100000	
Solidus Temperature (k)	constant   Edit	
	1150	
Liquidus Temperature (k)	constant	
	1150 +	
	Change/Create Delete Close Help	

- g. Enter 680 j / kg k for **Cp (Specific Heat)**.
- h. Enter 30 w/m-k for **Thermal Conductivity**.
- i. Enter 0.00553 kg/m-s for **Viscosity**.
- j. Enter 100000 j / kg for **Pure Solvent Melting Heat**.

Scroll down the group box to find **Pure Solvent Melting Heat** and the properties that follow.

- k. Enter 1150 *K* for **Solidus Temperature**.
- I. Enter 1150 *K* for **Liquidus Temperature**.
- m. Click Change/Create and close the Create/Edit Materials dialog box.

## 20.4.6. Setting the Cell Zone Conditions

1. Set the cell zone conditions for the fluid (**fluid**).

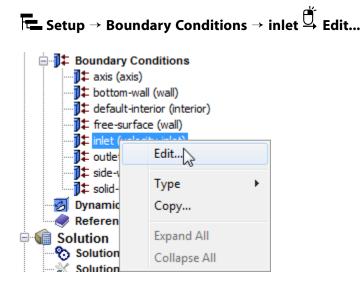
E Setup  $\rightarrow$  Cell Zone Conditions  $\rightarrow$  fluid  $\stackrel{\bullet}{\coprod}$  Edit...

E Fluid								×	
Zone Name fluid									
Material Name liquid-	metal	▼ Edit							
Frame Motion	Source Terms								
Mesh Motion	Fixed Values								
Porous Zone	_								
Reference Frame	Mesh Motion	Porous Zone	3D Fan Zone	Embedded LES	Reaction	Source Terms	Fixed Values	Multiphase	
	OK Cancel Heb								

- a. Select liquid-metal from the Material Name drop-down list.
- b. Click **OK** to close the **Fluid** dialog box.

# 20.4.7. Setting the Boundary Conditions

1. Set the boundary conditions for the inlet (**inlet**).



	Velocity Inlet							×	
	Zone Name								
	inlet								
	Momentum	Thermal	Radiation	Species	DPM	Multiphase	Potential	UDS	
	Velocity Specification Method Magnitude, Normal to Boundary								
		Velocity Ma	gnitude (m/s	) 0.00101		cons	tant	•	
	Supersonic/Init	ial Gauge Pre	essure (pascal	) 0		cons	tant	•	
			0	K Cancel	Help				

- a. Enter 0.00101 *m*/ *s* for **Velocity Magnitude**.
- b. Click the **Thermal** tab and enter 1300 *K* for **Temperature**.

Velocity Inlet		
Zone Name inlet		
Momentum Thermal	Radiation Species DPM Multiphase Potential UDS	_
Temperature (k) 1300	constant	
	OK Cancel Help	

- c. Click OK to close the Velocity Inlet dialog box.
- 2. Set the boundary conditions for the outlet (**outlet**).

# **E** Setup $\rightarrow$ Boundary Conditions $\rightarrow$ outlet $\stackrel{\text{D}}{\rightarrow}$ Edit...

Here, the solid is pulled out with a specified velocity, so a velocity inlet boundary condition is used with a positive axial velocity component.

Velocity Inlet	:				<b>—X</b> —			
Zone Name outlet								
Momentum	Thermal Radiation	Species DPM	Multiphase	Potential	UDS			
				POLEIILIAI				
Velocit	ty Specification Method Con	mponents			<b></b>			
	Reference Frame Ab	solute			<b></b>			
Supersonic/Init	tial Gauge Pressure (pascal)	0	СО	nstant	<b>_</b>			
	Axial-Velocity (m/s)	0.001	СО	nstant	<b>_</b>			
	Radial-Velocity (m/s)	0	СО	constant 🔹				
	Swirl-Velocity (m/s)	0	СО	nstant				
	Swirl Angular Velocity (rad/s) 1							
L	OK Cancel Help							

a. From the Velocity Specification Method drop-down list, select Components.

The **Velocity Inlet** dialog box will change to show related inputs.

- b. Enter 0.001 *m*/*s* for **Axial-Velocity**.
- c. Enter 1 *rad / s* for **Swirl Angular Velocity**.
- d. Click the **Thermal** tab and enter 500 *K* for **Temperature**.

💶 Velocity Inlet	t						×
Zone Name							
outlet		-					
Momentum	Thermal	Radiation	Species	DPM	Multiphase	Potential	UDS
Temperature (	k) 500		consta	int	•		
		0	K Cancel	Help			

- e. Click OK to close the Velocity Inlet dialog box.
- 3. Set the boundary conditions for the bottom wall (bottom-wall).



a. Click the **Thermal** tab.

🖬 Wall										×
Zone Name					_					
bottom-wall										
Adjacent Cell Zone fluid										
Momentum Thermal	Radiation	Species	DPM	Mu	ltiphase	UDS	Wall F	Film	Potential	
Thermal Conditions										
Heat Flux		т	Temperature	(k)	1300			cons	tant	•
Temperature					Wall	Thickness	(m) 0			P
<ul> <li>Convection</li> <li>Radiation</li> </ul>	He	at Generatio	on Rate (w/	m3)	0			cons	tant	•
<ul> <li>Mixed</li> </ul>	0	ontact Resist	tance (m2-)	(w)	0			cons	tant	-
via System Coupling										
via Mapped Interface										
Material Name										
aluminum 🔻	Edit									
		0	OK Canc	el	Help					

- i. Select Temperature in the Thermal Conditions group box.
- ii. Enter 1300 *K* for **Temperature**.
- b. Click **OK** to close the **Wall** dialog box.
- 4. Set the boundary conditions for the free surface (free-surface).

<b>F</b> Setup $\rightarrow$ Boundary Conditions $\rightarrow$ free-surface $\stackrel{\bigcup}{\rightarrow}$ Edit
--

The specified shear and Marangoni stress boundary conditions are useful in modeling situations in which the shear stress (rather than the motion of the fluid) is known. A free surface condition is an example of such a situation. In this case, the convection is driven by the Marangoni stress and the shear stress is dependent on the surface tension, which is a function of temperature.

💶 Wall
Zone Name
free-surface
Adjacent Cell Zone
fluid
Momentum Thermal Radiation Species DPM Multiphase UDS Wall Film Potential
Wall Motion       Motion         Image: Stationary Wall       Image: Wall Wall         Image: Moving Wall       Image: Wall Wall Wall
Shear Condition       Marangoni Stress         No Slip       Surface Tension Gradient (n/m-k) -0.00036         Specified Shear       Surface Tension Gradient (n/m-k) -0.00036         Marangoni Stress       Marangoni Stress
Wall Roughness         Roughness Height (m)         0         constant         Roughness Constant         0.5
OK Cancel Help

a. Select Marangoni Stress in the Shear Condition group box.

The **Marangoni Stress** condition allows you to specify the gradient of the surface tension with respect to temperature at a wall boundary.

- b. Enter -0.00036 n/m-k for Surface Tension Gradient.
- c. Click the **Thermal** tab to specify the thermal conditions.

Wall Zone Name free-surface Adjacent Cell Zo fluid	ne									<b>×</b>
Momentum	Thermal	Radiation	Species	DPM	Multip	hase	UDS	Wall Film	Potential	
Thermal Condi										
<ul> <li>Heat Flux</li> <li>Temperat</li> <li>Convection</li> <li>Radiation</li> </ul>	ure	He	at Transfer ( Free Stre	am Tempera		1500	/all Thicknes	[00	nstant nstant	• • •
<ul> <li>Mixed</li> <li>via System</li> </ul>	n Coupling			aration Rate					nstant	•
	d Interface		Contact F	Resistance (r	m2-k/w)	0			nstant	•
Material Name aluminum	•	Edit								
				OK Can	cel He	lp				

- i. Select **Convection** from the **Thermal Conditions** group box.
- ii. Enter 100  $w / m^2 k$  for Heat Transfer Coefficient.
- iii. Enter 1500 *K* for **Free Stream Temperature**.
- d. Click **OK** to close the **Wall** dialog box.
- 5. Set the boundary conditions for the side wall (side-wall).

**T** Setup  $\rightarrow$  Boundary Conditions  $\rightarrow$  side-wall  $\stackrel{0}{\hookrightarrow}$  Edit...

a. Click the **Thermal** tab.

Wall Zone Name side-wall Adjacent Cell Zone fluid										×
Momentum Thermal	Radiation	Species	DPM	Mu	ltiphase	UDS	Wall	Film	Potential	
Thermal Conditions Heat Flux		1	Temperature	e (k)	1400			cons	tant	•
Temperature     Convection	He	at Generati	on Rate (w/	/m3)		Thickness	(m) 0	cons	tant	P 
Radiation     Mixed     via System Coupling     via Mapped Interface	C	ontact Resis						cons	tant	•
Material Name aluminum	Edit									
		(	OK Canc	el	Help					

- i. Select Temperature from the Thermal Conditions group box.
- ii. Enter 1400 *K* for the **Temperature**.
- b. Click **OK** to close the **Wall** dialog box.
- 6. Set the boundary conditions for the solid wall (**solid-wall**).

**E** Setup  $\rightarrow$  Boundary Conditions  $\rightarrow$  solid-wall  $\stackrel{0}{\stackrel{\frown}{\rightarrow}}$  Edit...

🛂 Wall
Zone Name
solid-wall
Adjacent Cell Zone
fluid
Momentum Thermal Radiation Species DPM Multiphase UDS Wall Film Potential
Wall Motion Motion
<ul> <li>Stationary Wall</li> <li>● Relative to Adjacent Cell Zone Speed (rad/s) 0</li> <li>Constant ▼</li> <li>Absolute</li> </ul>
<ul> <li>Translational</li> <li>Rotational</li> <li>Components</li> </ul>
Shear Condition  No Slip  Specified Shear  Specularity Coefficient  Marangoni Stress
- Wall Roughness
Roughness Height (m) 0 constant
Roughness Constant 0.5 constant 👻
OK Cancel Help

a. From the Wall Motion group box, select Moving Wall.

The **Wall** dialog box is expanded to show additional parameters.

b. in the Motion group box, in the lower box, select Rotational.

The **Wall** dialog box is changed to show the rotational speed.

- c. Enter 1.0 *rad / s* for **Speed**.
- d. Click the **Thermal** tab to specify the thermal conditions.

🔜 Wall										×
Zone Name					_					
solid-wall Adjacent Cell Zo										
fluid	ne									
Momentum	Thermal	Radiation	Species	DPM	Multiphase	UDS	Wall Film	Potential		
Thermal Condit	ions									
			Te	emperature (k)	500		constant		-	
Temperati					Wall Ti	hickness (m) (	)		Р	
<ul> <li>Convection</li> <li>Radiation</li> </ul>	п	Hea	at Generatio	n Rate (w/m3)	0		constant		•	
Mixed		Co	ntact Resist	ance (m2-k/w)	0		constant		•	
<ul> <li>via System</li> <li>via Mappe</li> </ul>										
Material Name aluminum	•	Edit								
				OK Cance	l) Help					

- i. Select Temperature from the Thermal Conditions selection list.
- ii. Enter 500 *K* for **Temperature**.
- e. Click **OK** to close the **Wall** dialog box.

# 20.4.8. Solution: Steady Conduction

In this step, you will specify the discretization schemes to be used and temporarily disable the calculation of the flow and swirl velocity equations, so that only conduction is calculated. This steady-state solution will be used as the initial condition for the time-dependent fluid flow and heat transfer calculation.

1. Set the solution parameters.



Solution Methods
Pressure-Velocity Coupling
Scheme
Coupled
Spatial Discretization
Gradient
Least Squares Cell Based 🔹
Pressure
PRESTO!
Momentum
Second Order Upwind
Swirl Velocity
Second Order Upwind
Energy
Second Order Upwind
Transient Formulation
Non-Iterative Time Advancement
Frozen Flux Formulation
Pseudo Transient
Warped-Face Gradient Correction
High Order Term Relaxation Options
Default

- a. Select **Coupled** from the **Scheme** drop-down list in the **Pressure-Velocity Coupling** group box.
- b. Select PRESTO! from the Pressure drop-down list in the Spatial Discretization group box.

The **PRESTO!** scheme is well suited for rotating flows with steep pressure gradients.

- c. Retain the default selection of **Second Order Upwind** from the **Momentum**, **Swirl Velocity**, and **Energy** drop-down lists.
- d. Enable Pseudo Transient.

The Pseudo Transient option enables the pseudo transient algorithm in the coupled pressure-based solver. This algorithm effectively adds an unsteady term to the solution equations in order to improve stability and convergence behavior. Use of this option is recommended for general fluid flow problems.

2. Enable the calculation for energy.

Solving  $\rightarrow$  Controls  $\rightarrow$  Equations...

Equations	×
Equations [1/3]	
Flow Swirl Velocity	
Energy	
OK Default	Cancel Help

- a. Deselect **Flow** and **Swirl Velocity** from the **Equations** selection list to disable the calculation of flow and swirl velocity equations.
- b. Click **OK** to close the **Equations** dialog box.
- 3. Confirm the **Relaxation Factors**.

**Solving**  $\rightarrow$  Controls  $\rightarrow$  Controls...

Solution Controls	
Pseudo Transient Explicit Relaxation Factors	
Pressure	*
0.5	
Momentum	
0.5	Ξ
Density	
1	
Body Forces	
1	
Swirl Velocity	
0.75	
F	Ŧ
Default	
Equations Limits Advanced	

Retain the default values.

4. Enable the plotting of residuals during the calculation.

Solving  $\rightarrow$  Reports  $\rightarrow$  Residuals...

Residual Monitors					×
Options Print to Console Plot	Equations Residual energy	Monito	r Check Converg	ence Absolute Criteria	
Window 1 👻 Curves Axes					
Iterations to Plot 1000	Residual Values		Iterations	Convergence Criterion absolute	•
Iterations to Store	Compute Loca	I Scale	5 🐳	Convergence Condition	15
OK	Plot Renormali	ze Ca	ncel Help		.1

a. Ensure **Plot** is enabled in the **Options** group box.

Solving  $\rightarrow$  Initialization

- b. Click **OK** to accept the remaining default settings and close the **Residual Monitors** dialog box.
- 5. Initialize the solution.

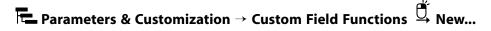
10001 30

	Initializ	ation	
Method		Patch	
O Hybrid	More Settings	Reset Statistics	
Standard	Options	Reset Statistics	t = 0
_	operation	Reset DPM	Initialize

a. Retain the **Method** at the default of **Hybrid** in the **Initialization** group.

For flows in complex topologies, hybrid initialization will provide better initial velocity and pressure field than standard initialization. This in general will help in improving the convergence behavior of the solver.

- b. Click Initialize.
- 6. Define a custom field function for the swirl pull velocity.



In this step, you will define a field function to be used to patch a variable value for the swirl pull velocity in the next step. The swirl pull velocity is equal to  $\Omega \cdot r$ , where  $\Omega$  is the angular velocity, and r is the radial coordinate. Since  $\Omega = 1$  rad/s, you can simplify the equation to simply r. In this example, the value of  $\Omega$ is included for demonstration purposes.

+	-	X	<i> </i>	y^x	ABS	Select Operand Field Functions from
INV	sin	COS	tan	h	log10	Field Functions Mesh
0	1	2	3	4	SQRT	Radial Coordinate
5	6	7	8	9	CE/C	Select
(	)	PI	e		DEL	Selecc

- a. From the Field Functions drop-down lists, select Mesh... and Radial Coordinate.
- b. Click the Select button to add radial-coordinate in the Definition field.

If you make a mistake, click the **DEL** button on the calculator pad to delete the last item you added to the function definition.

- c. Click the × button on the calculator pad.
- d. Click the 1 button.
- e. Enter omegar for New Function Name.
- f. Click Define.

The omegar item appears under the Parameters & Customisation/Parameters tree branch.

#### Note

To check the function definition or delete the custom field function, click **Manage...**. Then in the **Field Function Definitions** dialog box, from the **Field Functions** selection list, select **omegar** to view the function definition.

#### g. Close the Custom Field Function Calculator dialog box.

7. Patch the pull velocities.

#### Solving $\rightarrow$ Initialization $\rightarrow$ Patch...

As noted earlier, you will patch values for the pull velocities, rather than having ANSYS Fluent compute them. Since the radial pull velocity is zero, you will patch just the axial and swirl pull velocities.

Reference Frame <ul> <li>Relative to Cell Zone</li> <li>Absolute</li> </ul> /ariable           Pressure           Axial Velocity           Radial Velocity           Swirl Velocity           Temperature           Axial Pull Velocity           Radial Pull Velocity           Swirl Pull Velocity           Phi for wall distance	Value (m/s) 0.001 Use Field Function Field Function omegar	Zones to Patch Filter Text  Fluid  Registers to Patch [0/0]	
	Patch Clo	ose Help	

- a. From the Variable selection list, select Axial Pull Velocity.
- b. Enter 0.001 m/s for Value.
- c. From the **Zones to Patch** selection list, select **fluid**.
- d. Click Patch.

You have just patched the axial pull velocity. Next you will patch the swirl pull velocity.

Patch			×
Reference Frame <ul> <li>Relative to Cell Zone</li> <li>Absolute</li> </ul> Variable           Pressure           Axial Velocity           Radial Velocity           Swirl Velocity           Temperature           Axial Pull Velocity           Radial Pull Velocity           Pressure           Axial Pull Velocity           Phi for wall distance	Value (m/s) 0 Value Tield Function Field Function Omegar	Zones to Patch Filter Text  fluid  Registers to Patch [0/0]	
	Patch Clos	e Help	

e. From the Variable selection list, select Swirl Pull Velocity.

Scroll down the list to find **Swirl Pull Velocity**.

f. Enable the **Use Field Function** option.

- g. Select omegar from the Field Function selection list.
- h. Ensure that **fluid** is selected from the **Zones to Patch** selection list.
- i. Click Patch and close the Patch dialog box.
- 8. Save the initial case and data files (solid0.cas.gz and solid0.dat.gz).

**File**  $\rightarrow$  Write  $\rightarrow$  Case & Data...

9. Start the calculation by requesting 20 iterations.

### **Solving** $\rightarrow$ Run Calculation $\rightarrow$ Advanced...

Run Calculation	
Check Case	Update Dynamic Mesh
– Pseudo Transient Opti – Fluid Time Scale	ons
<ul> <li>Time Step Method</li> <li>● User Specified</li> <li>○ Automatic</li> </ul>	Pseudo Time Step (s)
Solid Time Scale Time Step Method © User Specified © Automatic	Pseudo Time Step (s) 1000
Number of Iterations	Reporting Interval
20	1
Profile Update Interval	
Data File Quantities	Acoustic Signals
Calculate	
Help	

- a. In the **Run Calculation** task page, select **User Specified** for the **Time Step Method** in both the **Fluid Time Scale** and the **Solid Time Scale** group boxes.
- b. Retain the default values of 1 and 1000 for the **Pseudo Time Step (s)** in the **Fluid Time Scale** and the **Solid Time Scale** group boxes, respectively.
- c. Enter 20 for Number of Iterations.
- d. Click Calculate.

The solution will converge in approximately 12 iterations.

10. Create and display the definition of filled temperature contours (Figure 20.3: Contours of Temperature for the Steady Conduction Solution (p. 832)).

Contours			×
Contour Name			
temperature			
Options	Contours of		
Filled	Temperature		•
Node Values	Static Temperature		•
<ul> <li>Global Range</li> <li>Auto Range</li> </ul>	Min	Max	
Clip to Range	0	0	
Draw Profiles	Surfaces Filter Text	<b></b>	<b>x</b> -
Coloring Banded Smooth Colormap Options	axis bottom-wall default-interior free-surface inlet outlet 		
Save/Display Compute Close Help			

Postprocessing  $\rightarrow$  Graphics  $\rightarrow$  Contours  $\rightarrow$  New...

- a. Enter temperature for Contour Name.
- b. Enable the **Filled** option.
- c. Select Temperature... and Static Temperature from the Contours of drop-down lists.
- d. Click Save/Display (Figure 20.3: Contours of Temperature for the Steady Conduction Solution (p. 832)).

The temperature contour definition appear under the Results/Graphics/Contours tree branch.

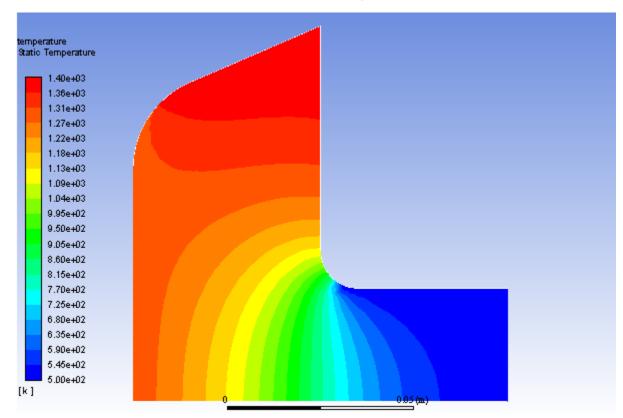


Figure 20.3: Contours of Temperature for the Steady Conduction Solution

11. Display filled contours of temperature to determine the thickness of mushy zone.

Postprocessing  $\rightarrow$  Graphics  $\rightarrow$  Contours  $\rightarrow$  Edit...

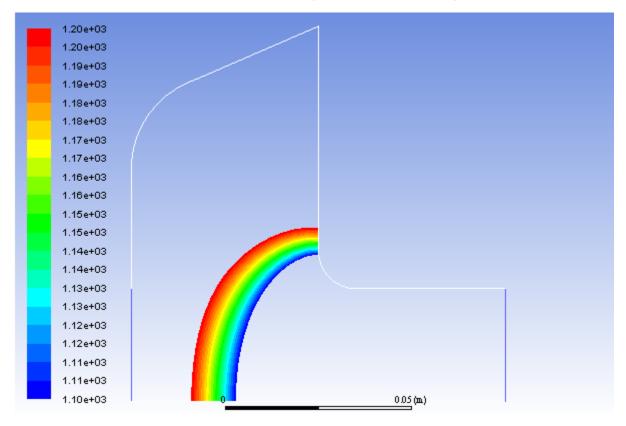
Contours			×
Options Filled	Contours of Temperature		•
Node Values	Static Temperature		•
Global Range Auto Range	Min (k)	Max (k)	_
Clip to Range	1100	1200	
Draw Profiles Draw Mesh	Surfaces Filter Text	-0	
Coloring Banded Smooth Levels Setup 20 1	axis bottom-wall default-interior free-surface inlet outlet side-wall New Surface Display Compute	Close Help	

a. Disable Auto Range in the Options group box.

The **Clip to Range** option is automatically enabled.

- b. Enter 1100 for **Min** and 1200 for **Max**.
- c. Click **Display** (See Figure 20.4: Contours of Temperature (Mushy Zone) for the Steady Conduction Solution (p. 833)) and close the **Contours** dialog box.

Figure 20.4: Contours of Temperature (Mushy Zone) for the Steady Conduction Solution



12. Save the case and data files for the steady conduction solution (solid.cas.gz and solid.dat.gz).

**File**  $\rightarrow$  Write  $\rightarrow$  Case & Data...

# 20.4.9. Solution: Transient Flow and Heat Transfer

In this step, you will turn on time dependence and include the flow and swirl velocity equations in the calculation. You will then solve the transient problem using the steady conduction solution as the initial condition.

1. Enable a time-dependent solution by selecting **Transient** from the **Time** list.

General		
Mesh		
Scale	Check	Report Quality
Display		
Solver		
Туре	Velo	city Formulation
Pressure-Base	sed 💿	Absolute
Density-Base	ed 💿	Relative
Time	2D 9	Space
Steady		Planar
Transient	$\odot$	Axisymmetric
	۲	Axisymmetric Swirl
Gravity	Units	]
Gravitational Ac	celeration	
X (m/s2) -9.81		P
Y (m/s2) 0		P
Z (m/s2) 0		P
Help		

2. Set the solution parameters.



Solution Methods	
Pressure-Velocity Coupling	
Scheme	
Coupled	•
Spatial Discretization	
Gradient	*
Least Squares Cell Based 🗸	
Pressure	
PRESTO!	
Momentum	Ξ
Second Order Upwind 👻	-
Swirl Velocity	
Second Order Upwind 👻	
Energy	
Second Order Upwind	-
Transient Formulation	
First Order Implicit	
Non-Iterative Time Advancement	
Frozen Flux Formulation	
Warped-Face Gradient Correction	
High Order Term Relaxation Options	
Default	

- a. Retain the default selection of First Order Implicit from the Transient Formulation drop-down list.
- b. Ensure that **PRESTO!** is selected from the **Pressure** drop-down list in the **Spatial Discretization** group box.
- 3. Enable calculations for flow and swirl velocity.



Equations	<b>X</b>
Equations [3/3]	
Flow Swirl Velocity Energy	
OK Default	Cancel Help

- a. Select Flow and Swirl Velocity and ensure that Energy is selected from the Equations selection list.
   Now all three items in the Equations selection list will be selected.
- b. Click **OK** to close the **Equations** dialog box.
- 4. Set the Under-Relaxation Factors.

 $\blacksquare Solving \rightarrow Controls \rightarrow Controls...$ 

Solut	tion Contro	ls	
Flow	Courant Num	ber	
200			
	– Explicit Rela	axation Factors	
	Momentum	0.75	
	Pressure	0.75	
Unde	r-Relaxation F	actors	_
De	ensity		
1			
Body Forces			
1			
Swirl Velocity			=
0.9			-
Liquid Fraction Update			
0.1			
En	ergy		
1			
			Ŧ
Defa	ault		
Equa	ations	nits Advanced	

- a. Enter 0.1 for Liquid Fraction Update.
- b. Retain the default values for other Under-Relaxation Factors.
- 5. Save the initial case and data files (solid01.cas.gz and solid01.dat.gz).

**File**  $\rightarrow$  Write  $\rightarrow$  Case & Data...

6. Run the calculation for 2 time steps.

**Solving**  $\rightarrow$  Run Calculation  $\rightarrow$  Advanced...

Run Calculation	
Check Case	Preview Mesh Motion
Time Stepping Method	Time Step Size (s)
Fixed •	0.1 P
Settings	Number of Time Steps
	2
Options	
Extrapolate Variables	
Data Sampling for Tin	ne Statistics
Sampling Interval	
1	Sampling Options
Time Sampled	(s) 0
Solid Time Step	
O User Specified	
Automatic	
Max Iterations/Time Step	Reporting Interval
20	1
Profile Update Interval	
1	
Data File Quantities	Acoustic Signals
Calculate	
Calcanded	
Help	

- a. Enter 0.1 s for **Time Step Size**.
- b. Set the Number of Time Steps to 2.
- c. Retain the default value of 20 for Max Iterations/Time Step.
- d. Click Calculate.
- 7. Display filled contours of the temperature after 0.2 seconds using the **temperature** contours definition that you created earlier.



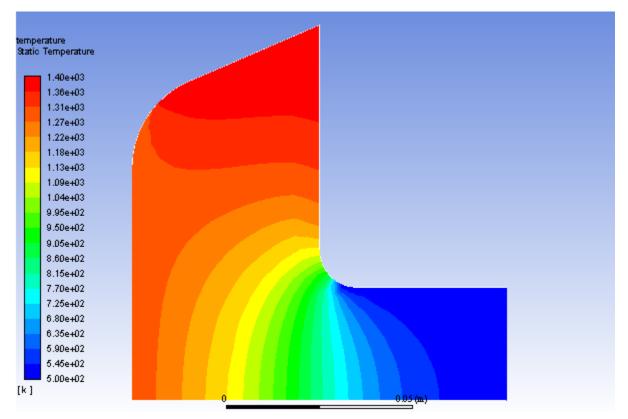


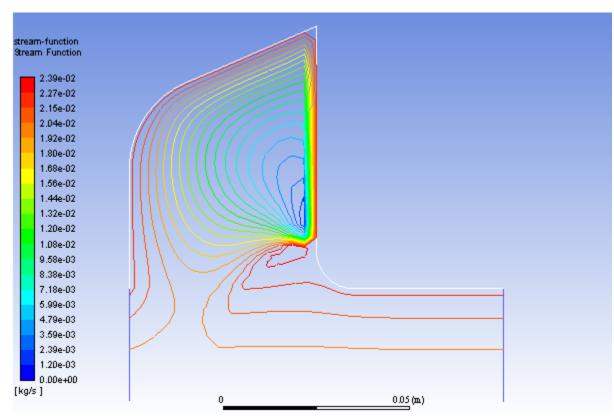
Figure 20.5: Contours of Temperature at t=0.2 s

8. Create and display the definition of stream function contours (Figure 20.6: Contours of Stream Function at t=0.2 s (p. 840)).



- a. Enter stream-function for Contour Name.
- b. Disable Filled in the Options group box.
- c. Select Velocity... and Stream Function from the Contours of drop-down lists.
- d. Click Save/Display.

The **stream-function** contour definition appear under the **Results/Graphics/Contours** tree branch.



#### Figure 20.6: Contours of Stream Function at t=0.2 s

As shown in Figure 20.6: Contours of Stream Function at t=0.2 s (p. 840), the liquid is beginning to circulate in a large eddy, driven by natural convection and Marangoni convection on the free surface.

9. Create and display the definition of liquid fraction contours by modifying the **stream-function** contour definition (Figure 20.7: Contours of Liquid Fraction at t=0.2 s (p. 841)).

Postprocessing  $\rightarrow$  Graphics  $\rightarrow$  Contours  $\rightarrow$  New...

- a. Enter liquid-fraction for Contour Name.
- b. Enable **Filled** in the **Options** group box.
- c. Select Solidification/Melting... and Liquid Fraction from the Contours of drop-down lists.
- d. Click **Save/Display** and close the **Contours** dialog box.

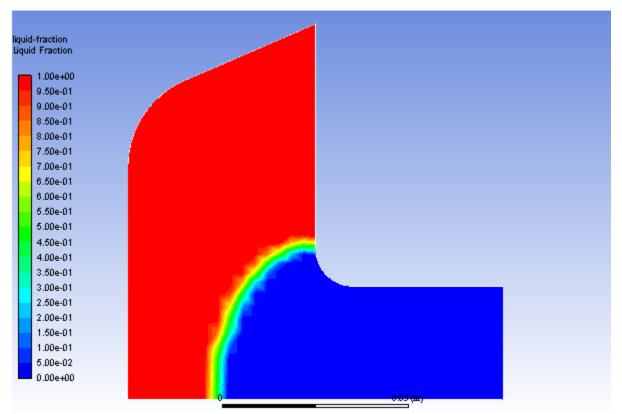


Figure 20.7: Contours of Liquid Fraction at t=0.2 s

The liquid fraction contours show the current position of the melt front. Note that in Figure 20.7: Contours of Liquid Fraction at t=0.2 s (p. 841), the mushy zone divides the liquid and solid regions roughly in half.

10. Continue the calculation for 48 additional time steps.



a. Enter 48 for Number of Time Steps.

#### b. Click Calculate.

After a total of 50 time steps have been completed, the elapsed time will be 5 seconds.

11. Display filled contours of the temperature after 5 seconds using the contour definition created earlier (Figure 20.8: Contours of Temperature at t=5 s (p. 842)).



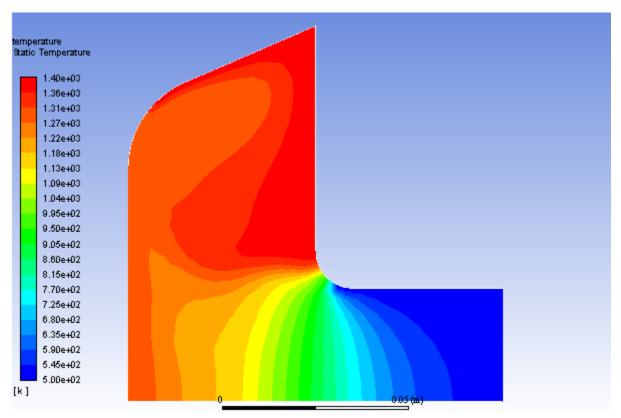


Figure 20.8: Contours of Temperature at t=5 s

As shown in Figure 20.8: Contours of Temperature at t=5 s(p. 842), the temperature contours are fairly uniform through the melt front and solid material. The distortion of the temperature field due to the recirculating liquid is also clearly evident.

In a continuous casting process, it is important to pull out the solidified material at the proper time. If the material is pulled out too soon, it will not have solidified (that is, it will still be in a mushy state). If it is pulled out too late, it solidifies in the casting pool and cannot be pulled out in the required shape. The optimal rate of pull can be determined from the contours of liquidus temperature and solidus temperature.

12. Display contours of stream function (Figure 20.9: Contours of Stream Function at t=5 s (p. 843)).

# **Results** $\rightarrow$ Graphics $\rightarrow$ Contours $\rightarrow$ stream-function $\stackrel{\frown}{\rightarrow}$ Display

As shown in Figure 20.9: Contours of Stream Function at t=5 s(p. 843), the flow has developed more fully by 5 seconds, as compared with Figure 20.6: Contours of Stream Function at t=0.2 s(p. 840) after 0.2 seconds. The main eddy, driven by natural convection and Marangoni stress, dominates the flow.

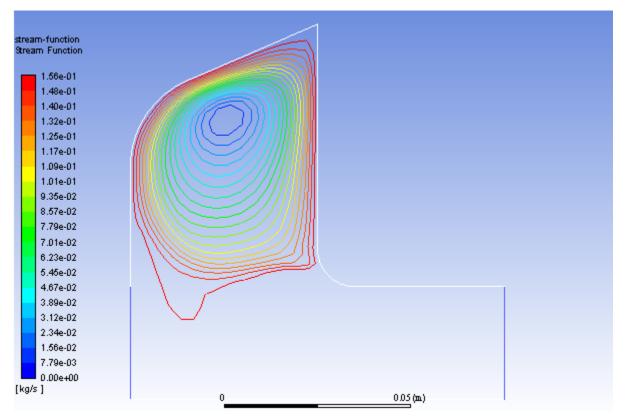


Figure 20.9: Contours of Stream Function at t=5 s

To examine the position of the melt front and the extent of the mushy zone, you will plot the contours of liquid fraction.

13. Display filled contours of liquid fraction (Figure 20.10: Contours of Liquid Fraction at t=5 s (p. 844)).

# **Results** $\rightarrow$ Graphics $\rightarrow$ Contours $\rightarrow$ liquid-fraction $\stackrel{\text{D}}{\rightarrow}$ Display

The introduction of liquid material at the left of the domain is balanced by the pulling of the solidified material from the right. After 5 seconds, the equilibrium position of the melt front is beginning to be established (Figure 20.10: Contours of Liquid Fraction at t=5 s(p. 844)).

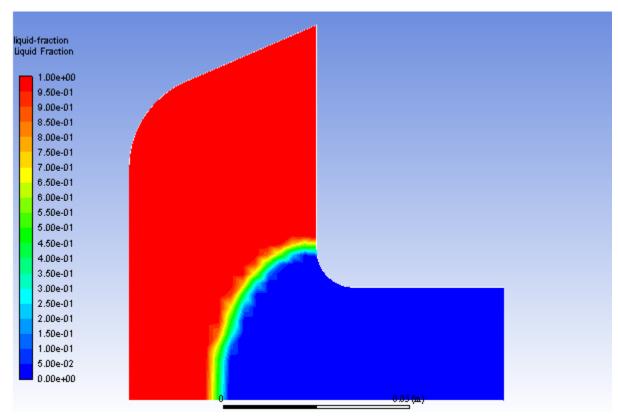


Figure 20.10: Contours of Liquid Fraction at t=5 s

14. Save the case and data files for the solution at 5 seconds (solid5.cas.gz and solid5.dat.gz).

```
File \rightarrow Write \rightarrow Case & Data...
```

# 20.5. Summary

In this tutorial, you studied the setup and solution for a fluid flow problem involving solidification for the Czochralski growth process.

The solidification model in ANSYS Fluent can be used to model the continuous casting process where a solid material is continuously pulled out from the casting domain. In this tutorial, you patched a constant value and a custom field function for the pull velocities instead of computing them. This approach is used for cases where the pull velocity is not changing over the domain, as it is computationally less expensive than having ANSYS Fluent compute the pull velocities during the calculation.

For more information about the solidification/melting model, see the Fluent User's Guide.

# 20.6. Further Improvements

This tutorial guides you through the steps to reach an initial set of solutions. You may be able to obtain a more accurate solution by using an appropriate higher-order discretization scheme and by adapting the mesh. Mesh adaption can also ensure that the solution is independent of the mesh. These steps are demonstrated in Introduction to Using ANSYS Fluent: Fluid Flow and Heat Transfer in a Mixing Elbow (p. 121).

# Chapter 21: Using the Eulerian Granular Multiphase Model with Heat Transfer

This tutorial is divided into the following sections:

21.1. Introduction21.2. Prerequisites21.3. Problem Description21.4. Setup and Solution21.5. Summary21.6. Further Improvements21.7. References

# 21.1. Introduction

This tutorial examines the flow of air and a granular solid phase consisting of glass beads in a hot gas fluidized bed, under uniform minimum fluidization conditions. The results obtained for the local wall-to-bed heat transfer coefficient in ANSYS Fluent can be compared with analytical results [1].

This tutorial demonstrates how to do the following:

- Use the Eulerian granular model.
- Set boundary conditions for internal flow.
- Compile a User-Defined Function (UDF) for the gas and solid phase thermal conductivities.
- Calculate a solution using the pressure-based solver.

# 21.2. Prerequisites

This tutorial is written with the assumption that you have completed one or more of the introductory tutorials found in this manual:

- Introduction to Using ANSYS Fluent in ANSYS Workbench: Fluid Flow and Heat Transfer in a Mixing Elbow (p. 1)
- Parametric Analysis in ANSYS Workbench Using ANSYS Fluent (p. 73)
- Introduction to Using ANSYS Fluent: Fluid Flow and Heat Transfer in a Mixing Elbow (p. 121)

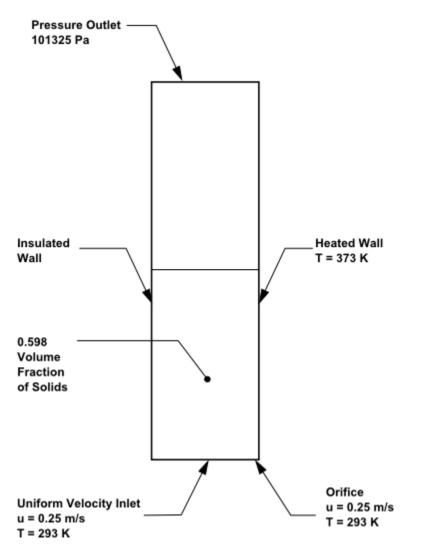
and that you are familiar with the ANSYS Fluent tree and ribbon structure. Some steps in the setup and solution procedure will not be shown explicitly.

In order to complete the steps to compile the UDF, you will need to have a working C compiler installed on your machine.

# 21.3. Problem Description

This problem considers a hot gas fluidized bed in which air flows upwards through the bottom of the domain and through an additional small orifice next to a heated wall. A uniformly fluidized bed is examined, which you can then compare with analytical results [1]. The geometry and data for the problem are shown in Figure 21.1: Problem Schematic (p. 846).

#### Figure 21.1: Problem Schematic



# 21.4. Setup and Solution

The following sections describe the setup and solution steps for this tutorial:

21.4.1. Preparation 21.4.2. Mesh 21.4.3. Solver Settings 21.4.4. Models 21.4.5. UDF 21.4.6. Materials 21.4.7. Phases 21.4.8. Boundary Conditions 21.4.9. Solution

#### 21.4.10. Postprocessing

#### 21.4.1. Preparation

To prepare for running this tutorial:

- 1. Set up a working folder on the computer you will be using.
- 2. Go to the ANSYS Customer Portal, https://support.ansys.com/training.

#### Note

If you do not have a login, you can request one by clicking **Customer Registration** on the log in page.

- 3. Enter the name of this tutorial into the search bar.
- 4. Narrow the results by using the filter on the left side of the page.
  - a. Click ANSYS Fluent under Product.
  - b. Click **18.0** under **Version**.
- 5. Select this tutorial from the list.
- 6. Click the eulerian\_granular\_heat\_R180.zip link to download the input files.
- 7. Unzip eulerian\_granular\_heat\_R180.zip to your working folder.

The files, fluid-bed.msh and conduct.c, can be found in the eulerian\_granular\_heat folder created after unzipping the file.

8. Use Fluent Launcher to enable **Double Precision** and start the **2D** version of ANSYS Fluent.

Fluent Launcher displays your **Display Options** preferences from the previous session.

For more information about the Fluent Launcher, see starting ANSYS Fluent using the Fluent Launcher in the Fluent Getting Started Guide.

- 9. Ensure that the Display Mesh After Reading and Workbench Color Scheme options are enabled.
- 10. Run in Serial by selecting Serial under Processing Options.
- 11. Ensure that **Set up Compilation Environment for UDF** is enabled in the **Environment** tab of the Fluent Launcher window. This will allow you to compile the UDF.

#### Note

The double precision solver is recommended for modeling multiphase flow simulations.

### 21.4.2. Mesh

1. Read the mesh file fluid-bed.msh.



As ANSYS Fluent reads the mesh file, it will report the progress in the console.

2. Check the mesh.

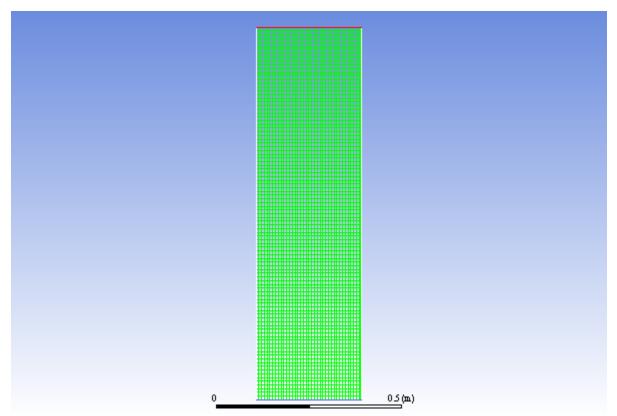
ANSYS Fluent will perform various checks on the mesh and will report the progress in the console. Make sure that the reported minimum volume is a positive number.

3. Examine the mesh (Figure 21.2: Mesh Display of the Fluidized Bed (p. 848)).

#### Extra

You can use the right mouse button to check which zone number corresponds to each boundary. If you click the right mouse button on one of the boundaries in the graphics window, its zone number, name, and type will be printed in the ANSYS Fluent console. This feature is especially useful when you have several zones of the same type and you want to distinguish between them quickly.

#### Figure 21.2: Mesh Display of the Fluidized Bed



### 21.4.3. Solver Settings

1. Enable the pressure-based transient solver.



General		
Mesh		
Scale	Check	Report Quality
Display		
Solver		
Туре	Veloc	ity Formulation
Pressure-Ba	sed 💿 A	Absolute
Density-Bas	ed 🔘 F	Relative
Time	2D S	pace
Steady		lanar
Transient	○ A	Axisymmetric
	© #	Axisymmetric Swirl
Constant of the second	(	
Gravity	Units	
Gravitational Ac	celeration	
X (m/s2) 0		P
Y (m/s2) -9.81		P
Z (m/s2) 0		Р
Help		

a. Retain the default selection of **Pressure-Based** from the **Type** list.

The pressure-based solver must be used for multiphase calculations.

- b. Select Transient from the Time list.
- 2. Set the gravitational acceleration.

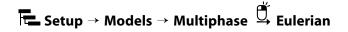
**Setup**  $\rightarrow$  **General**  $\rightarrow$  **Gravity** 

a. Enter -9.81 m/s<sup>2</sup> for the **Gravitational Acceleration** in the **Y** direction.

### 21.4.4. Models

1. Enable the Eulerian multiphase model for two phases.

You will use the default settings for the Eulerian model, so you can enable it directly from the tree by right-clicking the **Multiphase** node and choosing **Eulerian** from the context menu.



2. Enable heat transfer by enabling the energy equation.

```
\blacksquare Setup \to Models \to Energy \stackrel{\textcircled{}}{\to} On
```

An **Information** dialog box appears reminding you to confirm the property values. Click **OK** in the **Information** dialog box to continue.

3. Retain the default laminar viscous model.

The decision to use the laminar model should be based on the Stokes number for the particles suspended in the fluid flow.

**F**Setup  $\rightarrow$  Models  $\rightarrow$  Viscous  $\stackrel{\textcircled{0}}{\rightarrow}$  Model  $\rightarrow$  Laminar

### 21.4.5. UDF

1. Compile the user-defined function, conduct.c, that will be used to define the thermal conductivity for the gas and solid phase.

User Defined  $\rightarrow$  User Defined  $\rightarrow$  Functions  $\rightarrow$  Compiled...

Compiled UDFs		<b>X</b>
Source Files [0/1]	Header Files	<b>x</b> -
conduct.c		
Add Delete Library Name libudf	Add Delete	Build
	Load Cancel Help	h.

- a. Click the Add... button below the Source Files option to open the Select File dialog box.
  - i. Select the file **conduct.c** and click **OK** in the **Select File** dialog box.
- b. Click Build.

ANSYS Fluent will create a libudf folder and compile the UDF. Also, a **Warning** dialog box will open asking you to make sure that UDF source file and case/data files are in the same folder.

- c. Click **OK** to close the **Warning** dialog box.
- d. Click **Load** to load the UDF.

#### Extra

If you decide to read in the case file that is provided for this tutorial on the Customer Portal, you will need to compile the UDF associated with this tutorial in your working folder. This is necessary because ANSYS Fluent will expect to find the correct UDF libraries in your working folder when reading the case file.

#### 21.4.6. Materials

1. Modify the properties for air, which will be used for the primary phase.

**E** Setup 
$$\rightarrow$$
  $\bigcirc$  Materials  $\rightarrow \stackrel{\frown}{=}$  air  $\rightarrow$  Create/Edit...

The properties used for air are modified to match data used by Kuipers et al. [1]

Create/Edit Materials							×
Name		Material Type					Order Materials by
air		fluid				•	Name
Chemical Formula		Fluent Fluid Materials				_	Chemical Formula
		air				•	Fluent Database
		Mixture				_	User-Defined Database
		none				Ŧ	User-Denneu Database
Properties				_	_		
Density (kg/m3)	constant		▼ Edit		<u></u>		
	1.2						
Cp (Specific Heat) (j/kg-k)	constant		▼ Edit	.	=		
	994						
Thermal Conductivity (w/m-k)	user-defined		▼ Edit	JL	-		
	conduct_gas::ibu	udf					
Viscosity (kg/m-s)	constant		▼ Edit				
	1.7894e-05						
	r		<u>.</u>		<u> </u>		
	Chang	ge/Create Delete Close	Help				

- a. Enter 1.2 kg/m<sup>3</sup> for **Density**.
- b. Enter 994 J/kg-K for **Cp**.
- c. Select **user-defined** from the **Thermal Conductivity** drop-down list to open the **User Defined Functions** dialog box.
  - i. Select conduct\_gas::libudf from the available list.
  - ii. Click OK to close the User Defined Functions dialog box.
- d. Click Change/Create.

2. Define a new fluid material for the granular phase (the glass beads).

## Setup → $\bigcirc$ Materials → $\stackrel{\frown}{=}$ air → Create/Edit...

Create/Edit Materials					<b>×</b>
Name		Material Type			Order Materials by
solids		fluid		•	Name
Chemical Formula		Fluent Fluid Materials			Chemical Formula
		solids		•	Fluent Database
		Mixture			User-Defined Database
Bernardian		none		Ŧ	
Properties					
Density (kg/m3)	constant		Edit		
	2660				
Cp (Specific Heat) (j/kg-k)	constant		Edit		
	737				
Thermal Conductivity (w/m-k)	user-defined	,	Edit		
	conduct_solid::lit	budf			
Viscosity (kg/m-s)	constant	,	Edit		
	1.7894e-05				
Molecular Weight (kg/kmol)	constant	,	Edit	-	
	Chan	ge/Create Delete Close	Help		.ef

- a. Enter solids for Name.
- b. Enter 2660 kg/m<sup>3</sup> for **Density**.
- c. Enter 737 J/kg-K for **Cp**.
- d. Retain the selection of user-defined from the Thermal Conductivity drop-down list.
- e. Click the **Edit...** button to open the **User Defined Functions** dialog box.
  - i. Select conduct\_solid::libudf in the User Defined Functions dialog box and click OK.
     A Question dialog box will open asking if you want to overwrite air.
  - ii. Click **No** in the **Question** dialog box.
- f. Select solids from the Fluent Fluid Materials drop-down list.
- g. Click Change/Create and close the Materials dialog box.

#### 21.4.7. Phases

You will now configure the phases.

1. Define air as the primary phase.

```
ESetup \rightarrow Multiphase \rightarrow Phases \rightarrow phase-1 - Primary Phase \rightarrow Edit...
```

💶 Primary Phase		×
Name air		
Phase Material air	▼ Edit	
	OK Delete Cancel Help	

- a. Enter air for Name.
- b. Ensure that air is selected from the Phase Material drop-down list.
- c. Click **OK** to close the **Primary Phase** dialog box.
- 2. Define **solids** (glass beads) as the secondary phase.

**Setup**  $\rightarrow$  Multiphase  $\rightarrow$  Phases  $\rightarrow$  phase-2 - Secondary Phase  $\rightarrow$  Edit...

Name solids Phase Material solids Edit Granular Packed Bed Granular Temperature Model Phase Property Partial Differential Equation Properties Diameter (m) constant C Edit 0.0005 Granular Viscosity (kg/m-s) syamlal-obrien C Edit Granular Bulk Viscosity (kg/m-s) lun-et-al C Edit Frictional Viscosity (kg/m-s) none C Edit Granular Temperature (m2/s2) constant C Edit	<
Phase Material solids Edit Granular Packed Bed Granular Temperature Model Phase Property Partial Differential Equation Properties Diameter (m) constant Edit 0.0005 Granular Viscosity (kg/m-s) syamlal-obrien Edit Granular Bulk Viscosity (kg/m-s) lun-et-al Edit Frictional Viscosity (kg/m-s) none Edit	
<ul> <li>✓ Granular</li> <li>Packed Bed</li> <li>Granular Temperature Model</li> <li>● Phase Property</li> <li>● Partial Differential Equation</li> <li>Properties</li> <li>Diameter (m) constant          <ul> <li>■ Edit</li> <li>0.0005</li> <li>Granular Viscosity (kg/m-s) syamlal-obrien              <ul> <li>■ Edit</li> <li>■ Granular Bulk Viscosity (kg/m-s) lun-et-al                  <ul> <li>■ Edit</li> <li>■ Frictional Viscosity (kg/m-s) none                      <ul> <li>■ Edit</li> <li>■ Edit</li> </ul> <li>■ Edit</li> <li>■ Edit</li> <li>■ Edit</li> <li>■ Edit</li> <li>■ Edit</li> </li></ul> <li>■ Edit</li> <li>■ Edit</li> <li>■ Edit</li> </li></ul> <li>■ Edit</li> <li>■ Edit</li> <li>■ Edit</li> <li>■ Edit</li></li></ul></li></ul>	
<ul> <li>Packed Bed</li> <li>Granular Temperature Model</li> <li>Phase Property</li> <li>Partial Differential Equation</li> </ul> Properties           Diameter (m) constant         Edit           0.0005         Granular Viscosity (kg/m-s) syamlal-obrien           Granular Bulk Viscosity (kg/m-s) lun-et-al         Edit           Frictional Viscosity (kg/m-s) none         Edit	
Granular Temperature Model Phase Property Partial Differential Equation Properties Diameter (m) constant  Edit 0.0005 Granular Viscosity (kg/m-s) syamlal-obrien  Edit Granular Bulk Viscosity (kg/m-s) lun-et-al  Edit Frictional Viscosity (kg/m-s) none  Edit	
<ul> <li>Phase Property</li> <li>Partial Differential Equation</li> </ul> Properties           Diameter (m) constant         Edit           0.0005         Granular Viscosity (kg/m-s) syamlal-obrien         Edit           Granular Bulk Viscosity (kg/m-s) lun-et-al         Edit           Frictional Viscosity (kg/m-s) none         Edit	
<ul> <li>Partial Differential Equation</li> <li>Properties</li> <li>Diameter (m) constant          <ul> <li>Edit</li> <li>0.0005</li> <li>Granular Viscosity (kg/m-s) syamlal-obrien              <ul></ul></li></ul></li></ul>	
Properties Diameter (m) constant  Edit 0.0005 Granular Viscosity (kg/m-s) syamlal-obrien  Edit Granular Bulk Viscosity (kg/m-s) lun-et-al  Edit Frictional Viscosity (kg/m-s) none  Edit	
Diameter (m) constant       ▼ Edit         0.0005       Granular Viscosity (kg/m-s) syamlal-obrien       ▼ Edit         Granular Bulk Viscosity (kg/m-s) lun-et-al       ▼ Edit         Frictional Viscosity (kg/m-s) none       ▼ Edit	
0.0005 Granular Viscosity (kg/m-s) syamlal-obrien ▼ Edit Granular Bulk Viscosity (kg/m-s) lun-et-al ▼ Edit Frictional Viscosity (kg/m-s) none ▼ Edit	
Granular Viscosity (kg/m-s) syamlal-obrien   Edit Granular Bulk Viscosity (kg/m-s) lun-et-al  Frictional Viscosity (kg/m-s) none  Edit	
Granular Bulk Viscosity (kg/m-s) lun-et-al  Frictional Viscosity (kg/m-s) none Edit	
Frictional Viscosity (kg/m-s) none   Edit	
Frictional Viscosity (kg/m-s) none   Edit	
Granular Temperature (m2/s2) constant	
Granular Temperature (m2/s2) constant	
Euclide Comparation ( Compare )	
1e-05	
Solids Pressure (pascal) lun-et-al	
Radial Distribution lun-et-al	
Elasticity Modulus (pascal) derived   Edit	
Packing Limit constant   Edit	
0.6	
	_
OK Delete Cancel Help	

- a. Enter solids for Name.
- b. Select **solids** from the **Phase Material** drop-down list.
- c. Enable Granular.
- d. Retain the default selection of **Phase Property** in the **Granular Temperature Model** group box.
- e. Enter 0.0005 m for **Diameter**.

- f. Select syamlal-obrien from the Granular Viscosity drop-down list.
- g. Select lun-et-al from the Granular Bulk Viscosity drop-down list.
- h. Select **constant** from the **Granular Temperature** drop-down list and enter 1e-05.
- i. Enter 0.6 for the Packing Limit.
- j. Click **OK** to close the **Secondary Phase** dialog box.
- 3. Define the interphase interactions formulations to be used.

# **E** Setup $\rightarrow$ Multiphase $\rightarrow$ Phase Interactions $\stackrel{\bullet}{\coprod}$ Edit...

Phase Interaction			<b>×</b>				
Virtual Mass Drag	Lift Wall Lubrication	Turbulent Dispersion	Turbulence Interaction Colisto				
Drag Coefficient Drag Modification solids	air	syamlal-obrien	- Edit				
OK Cancel Help							

- a. In the Drag tab, select syamlal-obrien from the Drag Coefficient drop-down list.
- b. In the Heat tab, select gunn from the Heat Transfer Coefficient drop-down list.

The interphase heat exchange is simulated, using a drag coefficient, the default restitution coefficient for granular collisions of 0.9, and a heat transfer coefficient. Granular phase lift is not very relevant in this problem, and in fact is rarely used.

c. Click OK to close the Phase Interaction dialog box.

### 21.4.8. Boundary Conditions

For this problem, you need to set the boundary conditions for all boundaries.



Boundary Condi	tions	
Zone Filter Text	=0	=
default-interior		
poutlet		
v_jet		
v_uniform		
wall_hot		
wall_ins		
Phase Type	ID	
mixture 🔻 🗸 velo	city-inlet 🔻 5	]
Edit	Copy Profiles	
Parameters	Operating Conditions	
Display Mesh		
	Periodic Conditions	
Help		

1. Set the boundary conditions for the lower velocity inlet (**v\_uniform**) for the primary phase.

## **E** Setup $\rightarrow$ **\textcircled{}**Boundary Conditions $\rightarrow$ **\stackrel{\frown}{=}** v\_uniform

For the Eulerian multiphase model, you will specify conditions at a velocity inlet that are specific to the primary and secondary phases.

- a. Select air from the Phase drop-down list.
- b. Click the Edit... button to open the Velocity Inlet dialog box.

💶 Velocity Inlet	:						×
Zone Name					Pha	ase	_
v_uniform					air		
Momentum	Thermal	Radiation	Species	DPM	Multiphase	Potential	UDS
Velocity Specification Method Magnitude, Normal to Boundary							
Reference Frame Absolute							
Velocity Magnitude (m/s) 0.25 constant							
OK Cancel Help							

- i. Retain the default selection of **Magnitude**, **Normal to Boundary** from the **Velocity Specification Method** drop-down list.
- ii. Enter 0.25 m/s for the Velocity Magnitude.
- iii. Click the Thermal tab and enter 293 K for Temperature.
- iv. Click OK to close the Velocity Inlet dialog box.
- 2. Set the boundary conditions for the lower velocity inlet (**v\_uniform**) for the secondary phase.

## **E** Setup $\rightarrow$ **\bigcirc** Boundary Conditions $\rightarrow \stackrel{\frown}{=} v_{uniform}$

- a. Select **solids** from the **Phase** drop-down list.
- b. Click the Edit... button to open the Velocity Inlet dialog box.

Velocity Inlet			<b>—</b> ×—			
Zone Name v_uniform		Phase solids				
Momentum Therma	Radiation Species	DPM Multiphase	Potential UDS			
Temperature (k) 293	consta	nt 🔹				
OK Cancel Help						

- i. Retain the default Velocity Specification Method and Reference Frame.
- ii. Retain the default value of 0 m/s for the Velocity Magnitude.
- iii. Click the **Thermal** tab and enter 293 K for **Temperature**.

- iv. Click the Multiphase tab and retain the default value of 0 for Volume Fraction.
- v. Click **OK** to close the **Velocity Inlet** dialog box.
- 3. Set the boundary conditions for the orifice velocity inlet (**v\_jet**) for the primary phase.

E Setup  $\rightarrow$  Observed Boundary Conditions  $\rightarrow \stackrel{\bullet}{=} v_{jet}$ 

- a. Select air from the Phase drop-down list.
- b. Click the Edit... button to open the Velocity Inlet dialog box.

Velocity Inlet	:						×
Zone Name					Pha	ase	_
v_jet	-				air		
Momentum	Thermal	Radiation	Species	DPM	Multiphase	Potential	UDS
Velocity Specification Method Magnitude, Normal to Boundary  Reference Frame Absolute							]
Velocity Magnitude (m/s) 0.25							
OK Cancel Help							

- i. Retain the default Velocity Specification Method and Reference Frame.
- ii. Enter 0.25 m/s for the Velocity Magnitude.

In order for a comparison with analytical results [1] to be meaningful, in this simulation you will use a uniform value for the air velocity equal to the minimum fluidization velocity at both inlets on the bottom of the bed.

- iii. Click the **Thermal** tab and enter 293 K for **Temperature**.
- iv. Click **OK** to close the **Velocity Inlet** dialog box.
- 4. Set the boundary conditions for the orifice velocity inlet (**v\_jet**) for the secondary phase.

## **E** Setup $\rightarrow$ **Conditions** $\rightarrow$ **E** v\_jet

- a. Select solids from the Phase drop-down list.
- b. Click the Edit... button to open the Velocity Inlet dialog box.

💶 Velocity Inlet	:						×
Zone Name					Pha		_
v_jet					sol	ids	
Momentum	Thermal	Radiation	Species	DPM	Multiphase	Potential	UDS
Velocity Specification Method Magnitude, Normal to Boundary							
Reference Frame Absolute							
Velocity Magnitude (m/s) 0 constant							
OK Cancel Help							
							_

- i. Retain the default Velocity Specification Method and Reference Frame.
- ii. Retain the default value of 0 m/s for the Velocity Magnitude.
- iii. Click the **Thermal** tab and enter 293 K for **Temperature**.
- iv. Click the Multiphase tab and retain the default value of 0 for the Volume Fraction.
- v. Click OK to close the Velocity Inlet dialog box.
- 5. Set the boundary conditions for the pressure outlet (**poutlet**) for the mixture phase.

## **E** Setup $\rightarrow$ **Conditions** $\rightarrow$ **E** poutlet

For the Eulerian granular model, you will specify conditions at a pressure outlet for the mixture and for both phases.

The thermal conditions at the pressure outlet will be used only if flow enters the domain through this boundary. You can set them equal to the inlet values, as no flow reversal is expected at the pressure outlet. In general, however, it is important to set reasonable values for these downstream scalar values, in case flow reversal occurs at some point during the calculation.

- a. Select mixture from the Phase drop-down list.
- b. Click the Edit... button to open the Pressure Outlet dialog box.
  - i. Retain the default value of 0 Pascal for Gauge Pressure.
  - ii. Click **OK** to close the **Pressure Outlet** dialog box.
- 6. Set the boundary conditions for the pressure outlet (poutlet) for the primary phase.

### **E** Setup $\rightarrow$ **Conditions** $\rightarrow$ **E** poutlet

- a. Select **air** from the **Phase** drop-down list.
- b. Click the Edit... button to open the Pressure Outlet dialog box.

Pressure Outle	et						×
Zone Name					Pha	ase	_
poutlet					air		
Momentum	Thermal	Radiation	Species	DPM	Multiphase	Potential	UDS
Backflow Total	Temperature	e (k) 293			constant	•	
OK Cancel Help							

- i. In the Thermal tab, enter 293 K for Backflow Total Temperature.
- ii. Click **OK** to close the **Pressure Outlet** dialog box.
- 7. Set the boundary conditions for the pressure outlet (**poutlet**) for the secondary phase.

**E** Setup  $\rightarrow$  **\bigcirc** Boundary Conditions  $\rightarrow \stackrel{\frown}{=}$  poutlet

- a. Select **solids** from the **Phase** drop-down list.
- b. Click the **Edit...** button to open the **Pressure Outlet** dialog box.

Pressure Outle	et						×
Zone Name poutlet					Pha sol		
Momentum	Thermal	Radiation	Species	DPM	Multiphase	Potential	UDS
Backflow Total Temperature (k) 293 Constant							
OK Cancel Help							

- i. In the Thermal tab, enter 293 K for the Backflow Total Temperature.
- ii. In the Multiphase tab, retain default settings.
- iii. Click **OK** to close the **Pressure Outlet** dialog box.
- 8. Set the boundary conditions for the heated wall (wall\_hot) for the mixture.

**Setup**  $\rightarrow$  **\clubsuit Boundary Conditions**  $\rightarrow$  **\equiv** wall\_hot

For the heated wall, you will set thermal conditions for the mixture, and momentum conditions (zero shear) for both phases.

- a. Select mixture from the Phase drop-down list.
- b. Click the **Edit...** button to open the **Wall** dialog box.

Wall Zone Name wall_hot Adjacent Cell Zone fluid		Phase			×
Momentum Thermal Thermal Conditions Heat Flux Temperature Convection Radiation Mixed via System Coupling via Mapped Interface	DPM Mu Temperature (k) ion Rate (w/m3)	Wall Thick	ness (m) 0	Film Potential	• •
Material Name aluminum 👻	OK Cancel	Help			

- i. In the Thermal tab, select Temperature from the Thermal Conditions list.
- ii. Enter 373 K for Temperature.
- iii. Click **OK** to close the **Wall** dialog box.
- 9. Set the boundary conditions for the heated wall (**wall\_hot**) for the primary phase.

**E** Setup  $\rightarrow$  **\bigcirc** Boundary Conditions  $\rightarrow$  **\stackrel{\frown}{=}** wall\_hot

- a. Select air from the Phase drop-down list.
- b. Click the Edit... button to open the Wall dialog box.

💶 Wall								×
Zone Name					Ph	ase		
wall_hot					ai	r		
Adjacent Cell Zo	one							
fluid								
Momentum	Thermal	Radiation	Species	DPM	Multiphase	UDS	Wall Film	Potential
Shear Condit No Slip Specified 3 Specularity Marangoni	Shear y Coefficient							
			ОКС	ancel	Help			

- c. Retain the default **No Slip** condition and click **OK** to close the **Wall** dialog box.
- 10. Set the boundary conditions for the heated wall (**wall\_hot**) for the secondary phase (**solids**) same as that of the primary phase.



For the secondary phase, you will retain the default no slip condition as for the primary phase.

11. Set the boundary conditions for the adiabatic wall (**wall\_ins**).

**E** Setup 
$$\rightarrow$$
  **$\bigcirc$**  Boundary Conditions  $\rightarrow$   **$\equiv wall_ins$** 

For the adiabatic wall, retain the default thermal conditions for the mixture (zero heat flux), and the default momentum conditions (no slip) for both phases.

### 21.4.9. Solution

1. Select the second order implicit transient formulation and higher-order spatial discretization schemes.

Solving  $\rightarrow$  Solution  $\rightarrow$  Methods...

Solution Methods
Pressure-Velocity Coupling
Scheme
Phase Coupled SIMPLE
Solve N-Phase Volume Fraction Equations
Spatial Discretization
Gradient
Least Squares Cell Based
Momentum
Second Order Upwind
Volume Fraction
QUICK 🔹
Energy
QUICK 🔹
Transient Formulation
Second Order Implicit 🔹
Non-Iterative Time Advancement
Frozen Flux Formulation
Warped-Face Gradient Correction
High Order Term Relaxation Options
Default

a. Select Second Order Implicit from the Transient Formulation drop-down list.

- b. Modify the discretization methods in the **Spacial Discretization** group box.
  - i. Select Second Order Upwind for Momentum.
  - ii. Select Quick for Volume Fraction and Energy.
- 2. Set the solution parameters.

Solution Controls	
Under-Relaxation Factors	
Pressure	
0.5	
Density	
1	
Body Forces	
1	
Momentum	
0.2	
Volume Fraction	
0.5	
E	Ψ.
Default	
Equations Limits Advanced	

Solving  $\rightarrow$  Controls  $\rightarrow$  Controls...

- a. Enter 0.5 for **Pressure**.
- b. Enter 0.2 for Momentum.
- 3. Ensure that the plotting of residuals is enabled during the calculation.

**Solving**  $\rightarrow$  Reports  $\rightarrow$  Residuals...

4. Define a custom field function for the heat transfer coefficient.

User Defined  $\rightarrow$  Field Functions  $\rightarrow$  Custom...

Initially, you will define functions for the mixture temperature, and thermal conductivity, then you will use these to define a function for the heat transfer coefficient.

Definition	ield Function Ca		ture * enlideu	of		
+ INV 0 5 (		X COS 2 7 PI	/ 1000000000000000000000000000000000000	y^x h 4 9	ABS log10 SQRT CE/C DEL	Select Operand Field Functions from Field Functions Phases Volume fraction Phase Solids Select
New Functio	n Name t_mid		Defin	e Manage	Close H	telp

- a. Define the function t\_mix.
  - i. Select Temperature... and Static Temperature from the Field Functions drop-down lists.
  - ii. Ensure that air is selected from the Phase drop-down list and click Select.
  - iii. Click the multiplication symbol in the calculator pad.
  - iv. Select Phases... and Volume fraction from the Field Functions drop-down list.
  - v. Ensure that air is selected from the Phase drop-down list and click Select.
  - vi. Click the addition symbol in the calculator pad.
  - vii. Similarly, add the term solids-temperature \* solids-vof.
  - viii.Enter t\_mix for New Function Name.
  - ix. Click Define.
- b. Define the function k\_mix.

💶 Custom Fi	ield Function Ca	lculator								
Definition										
air-thermal-c	onductivity-lam	* air-vof + so	ids-thermal-co	onductivity-lan	n * solids-vof					
+		X	1	y^x	ABS	Select Operand Field Functions from				
INV	sin	COS	tan	h	log10	Field Functions				
0	1	2	3	4	SQRT					
5	6	7	8	9	CE/C	Volume fraction    Phase				
(		PI	e	Î.	DEL	solids				
					··	Select				
	Selecc									
New Function	n Name k_mix									
	Define Manage Close Help									

i. Select Properties... and Thermal Conductivity from the Field Functions drop-down lists.

- ii. Select air from the Phase drop-down list and click Select.
- iii. Click the multiplication symbol in the calculator pad.
- iv. Select Phases... and Volume fraction from the Field Functions drop-down lists.
- v. Ensure that air is selected from the Phase drop-down list and click Select.
- vi. Click the addition symbol in the calculator pad.
- vii. Similarly, add the term solids-thermal-conductivity-lam \* solids-vof.

viii.Enter k\_mix for New Function Name.

#### ix. Click Define.

c. Define the function <code>ave\_htc</code>.

	Custom Fiel	d Function Ca	lculator							
	efinition									
Ŀ	k_mix * (t_r	mix - 373) / (	58.5 * 10 ^ (	- 6)) / 80						
	+	-	X		у^х	ABS	Select Operand Field Functions from			
	INV	sin	cos	tan	h	log10	Field Functions			
	0	1	2	3	4	SQRT	t_mix •			
	5	6	7	8	9	CE/C	Phase			
	(	)	PI	е		DEL	mixture 🔻			
							Select			
		United and an	function D							
-	ew Function	Name custom	-function-3							
	Define Manage Close Help									

- i. Click the subtraction symbol in the calculator pad.
- ii. From the **Field Functions** drop-down lists, select **Custom Field Functions...** and **k\_mix** and click **Select**.
- iii. Use the calculator pad and the Field Functions lists to complete the definition of the function.

 $-k_{mix} \times (t_{mix} - 373) / (58.5 \times 10^{(-6)}) / 80$ 

- iv. Enter ave\_htc for New Function Name.
- v. Click **Define** and close the **Custom Field Function Calculator** dialog box.
- 5. Define the point surface in the cell next to the wall on the plane y=0.24.

Setting Up Domain  $\rightarrow$  Surface  $\rightarrow$  Create  $\rightarrow$  Point...

Point Surface	×							
Options Point Tool	Coordinates x0 (m) 0.28494							
Reset	y0 (m) 0.24 z0 (m) 0							
Sele	Select Point with Mouse							
New Surface Nar	ne							
y=0.24								
Create Manage Close Help								

- a. Enter 0.28494 m for **x0** and 0.24 m for **y0** in the **Coordinates** group box.
- b. Enter y=0.24 for **New Surface Name**.
- c. Click **Create** and close the **Point Surface** dialog box.
- 6. Create a surface report definition for the heat transfer coefficient.

**Solving**  $\rightarrow$  Reports  $\rightarrow$  Definitions  $\rightarrow$  New  $\rightarrow$  Surface Report  $\rightarrow$  Facet Average

Surface Report Definition	×
Name	Report Type
surf-mon-1	Facet Average
Options	Custom Vectors
	Vectors of
Per Surface	· · · · · · · · · · · · · · · · · · ·
Average Over	Custom Vectors
1	Field Variable
Report Files [0/0]	Custom Field Functions
	ave_htc
	Phase
	mixture 🔹
	Surfaces Filter Text
Report Plots [0/0]	default-interior
	poutlet
	v_jet
	v_uniform
	wall_hot wall_ins
Create	y=0.24
Report File	,
Report Plot	
Frequency 1	
Print to Console	
Create Output Parameter	New Surface 🔻
ОК Сотры	ute Cancel Help

- a. Enter **surf-mon-1** for **Name** of the surface report definition.
- b. In the **Create** group box, enable **Report File**, **Report Plot** and **Print to Console**.
- c. Select Custom Field Functions... and ave\_htc from the Field Variable drop-down lists.
- d. Select **y=0.24** from the **Surfaces** selection list.
- e. Click **OK** to save the surface report definition settings and close the **Surface Report Definition** dialog box.

**surf-mon-1-rplot** and **surf-mon-1-rfile** that are automatically generated by Fluent appear in the tree (under **Solution/Monitors/Report Plots** and **Solution/Monitors/Report Files**, respectively).

f. Rename the report output file.



Edit Report File Name			<u></u>
surf-mon-1-rfile			
Available Report Definitions [0/3]	x- 🤁	Selected Report Definitions [0/1]	x-1 🖓
delta-time flow-time iters-per-timestep		1>> emove	
Output File Base Name htc-024.out Full File Name Get Data Every 1 🔅 time-step V Print	Browse To Console	New V Edit	
	OK Can	cel Help	

- i. Enter htc-024.out for Output File Base Name.
- ii. Click **OK** to close the **Edit Report File** dialog box.
- 7. Initialize the solution.



Solution Initialization	
Initialization Methods	
Hybrid Initialization	
Standard Initialization	
Compute from	
all-zones 👻	
Reference Frame Relative to Cell Zone Absolute	
Initial Values	
Gauge Pressure (pascal)	-
0	
air X Velocity (m/s)	
0	
air Y Velocity (m/s)	E
0	
air Temperature (k)	
293	
solids X Velocity (m/s)	
0	
solids Y Velocity (m/s)	
0	
	Ŧ
Initialize Reset Patch	
Reset DPM Sources Reset Statistics	

- a. Select **all-zones** from the **Compute from** drop-down list.
- b. Retain the default values and click Initialize.
- 8. Define an adaption register for the lower half of the fluidized bed.

### $\blacksquare Setting Up Domain \rightarrow Adapt \rightarrow Mark/Adapt Cells \rightarrow Region...$

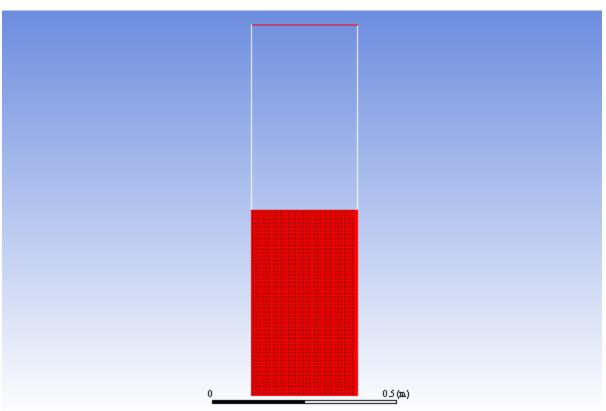
This register is used to patch the initial volume fraction of solids in the next step.

Region Adaption				
Options Inside	Input Coordina X Min (m)	X Max (m)		
<ul> <li>Outside</li> <li>Shapes</li> </ul>	0 Y Min (m)	0.3 Y Max (m)		
<ul> <li>Quad</li> <li>Circle</li> <li>Quinder</li> </ul>	0 Z Min (m)	0.5 Z Max (m)		
<ul> <li>Cylinder</li> <li>Manage</li> </ul>	0			
Controls Select Points with Mouse				
Adapt Mark Close Help				

- a. Enter 0.3 m for Xmax and 0.5 m for Ymax in the Input Coordinates group box.
- b. Click Mark.
- c. Click the Manage... button to open the Manage Adaption Registers dialog box.
  - i. Ensure that **hexahedron-r0** is selected from the **Registers** selection list.
  - ii. Click **Display** and close the **Manage Adaption Registers** dialog box.

After you define a region for adaption, it is a good practice to display it to visually verify that it encompasses the intended area.





- d. Close the **Region Adaption** dialog box.
- 9. Patch the initial volume fraction of solids in the lower half of the fluidized bed.

S S	olving →	Initialization	→ Patch
-----	----------	----------------	---------

Patch			<b>×</b>
Reference Frame Relative to Cell Zone Absolute Phase solids Variable X Velocity Y Velocity Temperature Granular Temperature Volume Fraction	Value 0.598 Use Field Function Field Function t_mix k_mix ave_htc	Zones to Patch Filter Text fluid Registers to Patch [1/1] hexahedron-r0	
Patch Close Help			

a. Select **solids** from the **Phase** drop-down list.

- b. Select Volume Fraction from the Variable selection list.
- c. Enter 0.598 for Value.
- d. Select hexahedron-r0 from the Registers to Patch selection list.
- e. Click Patch and close the Patch dialog box.

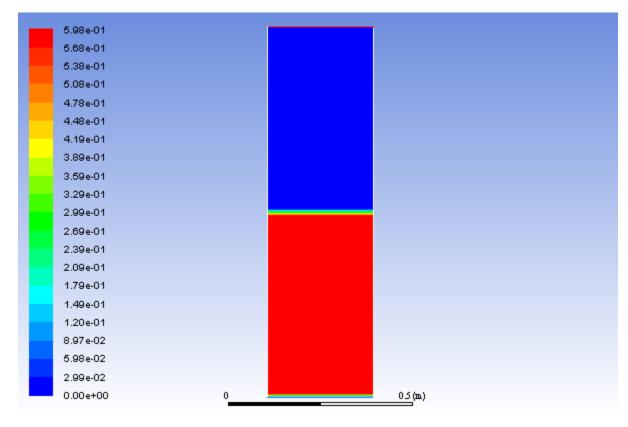
At this point, it is a good practice to display contours of the variable you just patched, to ensure that the desired field was obtained.

10. Display contours of **Volume Fraction** of solids (Figure 21.4: Initial Volume Fraction of Granular Phase (solids) (p. 872)).



- a. Enable Filled in the Options group box.
- b. Select Phases... from the upper Contours of drop-down list.
- c. Select **solids** from the **Phase** drop-down list.
- d. Ensure that Volume fraction is selected from the lower Contours of drop-down list.
- e. Click **Display** and close the **Contours** dialog box.

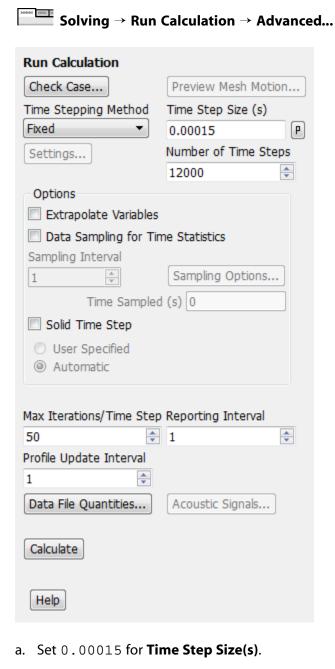




11. Save the case file (fluid-bed.cas.gz).



12. Start calculation.



- b. Set 12000 for **Number of Time Steps**.
- c. Enter 50 for Max Iterations/Time Step.
- d. Click Calculate.

The plot of the value of the mixture-averaged heat transfer coefficient in the cell next to the heated wall versus time is in excellent agreement with results published for the same case [1].

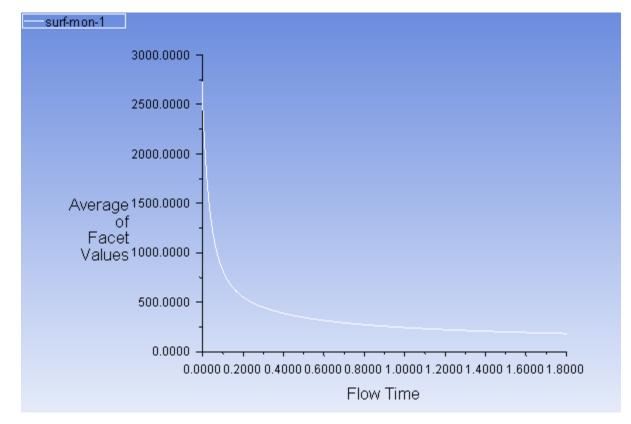


Figure 21.5: Plot of Mixture-Averaged Heat Transfer Coefficient in the Cell Next to the Heated Wall Versus Time

13. Save the case and data files (fluid-bed.cas.gz and fluid-bed.dat.gz).

**File**  $\rightarrow$  Write  $\rightarrow$  Case & Data...

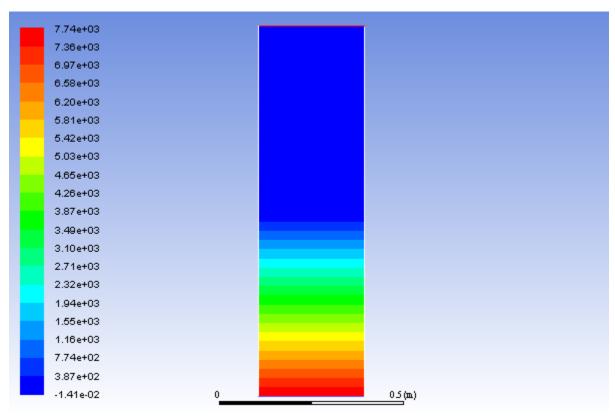
#### 21.4.10. Postprocessing

1. Display the pressure field in the fluidized bed (Figure 21.6: Contours of Static Pressure (p. 876)).

Postprocessing  $\rightarrow$  Graphics  $\rightarrow$  Contours  $\rightarrow$  Edit...

Contours	
Options Filled Options Options Filled Options Optio	Contours of Pressure  Static Pressure
Auto Range     Clip to Range	Phase mixture
Draw Profiles	Min (pascal) Max (pascal) -0.01413127 7744.819
Coloring Banded Smooth Levels Setup 20 1	Surfaces Filter Text
	Display Compute Close Help

- a. Select **mixture** from **Phase** drop-down list.
- b. Select Pressure... and Static Pressure from the Contours of drop-down lists.
- c. Click **Display**.



#### Figure 21.6: Contours of Static Pressure

Note the build-up of static pressure in the granular phase.

- 2. Display the volume fraction of solids (Figure 21.7: Contours of Volume Fraction of Solids (p. 877)).
  - a. Select **solids** from the **Phase** drop-down list.
  - b. Select Phases... and Volume fraction from the Contours of drop-down lists.
  - c. Click **Display** and close the **Contours** dialog box.
  - d. Zoom in to show the contours close to the region where the change in volume fraction is the greatest.

Note that the region occupied by the granular phase has expanded slightly, as a result of fluidization.

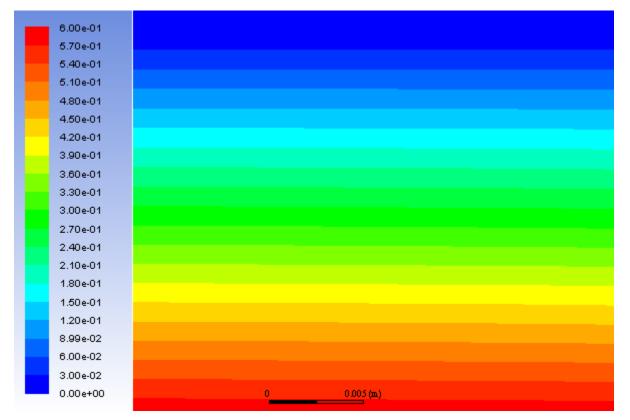


Figure 21.7: Contours of Volume Fraction of Solids

### 21.5. Summary

This tutorial demonstrated how to set up and solve a granular multiphase problem with heat transfer, using the Eulerian model. You learned how to set boundary conditions for the mixture and both phases. The solution obtained is in excellent agreement with analytical results from Kuipers et al. [1].

### 21.6. Further Improvements

This tutorial guides you through the steps to reach an initial solution. You may be able to obtain a more accurate solution by using an appropriate higher-order discretization scheme and by adapting the mesh further. Mesh adaption can also ensure that the solution is independent of the mesh. These steps are demonstrated in Introduction to Using ANSYS Fluent: Fluid Flow and Heat Transfer in a Mixing Elbow (p. 121).

## 21.7. References

1. J. A. M. Kuipers, W. Prins, and W. P. M. Van Swaaij "Numerical Calculation of Wall-to-Bed Heat Transfer Coefficients in Gas-Fluidized Beds", Department of Chemical Engineering, Twente University of Technology, in AIChE Journal, July 1992, Vol. 38, No. 7.

## **Chapter 22: Postprocessing**

This tutorial is divided into the following sections:

22.1. Introduction22.2. Prerequisites22.3. Problem Description22.4. Setup and Solution22.5. Summary

## 22.1. Introduction

This tutorial demonstrates the postprocessing capabilities of Fluent using a 3D model of a flat circuit board with a heat generating electronic chip mounted on it. The flow over the chip is laminar and involves conjugate heat transfer.

The heat transfer involves conduction in the chip and conduction and convection in the surrounding fluid. The physics of conjugate heat transfer such as this are common in many engineering applications, including the design and cooling of electronic components.

In this tutorial, you will read the case and data files (without doing the calculation) and perform a number of postprocessing exercises.

This tutorial demonstrates how to do the following:

- Add lights to the display at multiple locations.
- Create surfaces for the display of 3D data.
- Display filled contours of temperature on several surfaces.
- Display velocity vectors.
- Mirror a display about a symmetry plane.
- Create animations.
- Create a scene.
- Display results on successive slices of the domain.
- Display pathlines.
- Plot quantitative results.
- Overlay and "explode" a display.
- Annotate the display.

### 22.2. Prerequisites

This tutorial is written with the assumption that you have completed one or more of the introductory tutorials found in this manual:

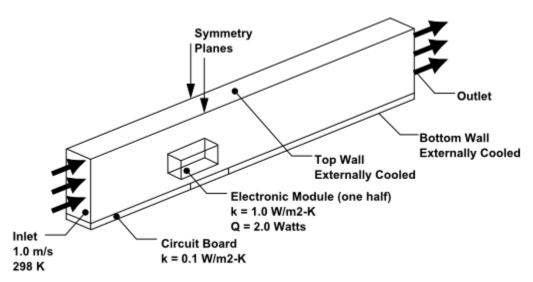
- Introduction to Using ANSYS Fluent in ANSYS Workbench: Fluid Flow and Heat Transfer in a Mixing Elbow (p. 1)
- Parametric Analysis in ANSYS Workbench Using ANSYS Fluent (p. 73)
- Introduction to Using ANSYS Fluent: Fluid Flow and Heat Transfer in a Mixing Elbow (p. 121)

and that you are familiar with the Fluent tree and ribbon structure. Some steps in the setup and solution procedure will not be shown explicitly.

## 22.3. Problem Description

The problem considered is shown schematically in Figure 22.1: Problem Specification (p. 880). The configuration consists of a series of side-by-side electronic chips, or modules, mounted on a circuit board. Air flow, confined between the circuit board and an upper wall, cools the modules. To take advantage of the symmetry present in the problem, the model will extend from the middle of one module to the plane of symmetry between it and the next module.

As shown in the figure, each half-module is assumed to generate 2.0 Watts and to have a bulk conductivity of 1.0 W/m<sup>2</sup>-K. The circuit board conductivity is assumed to be one order of magnitude lower:  $0.1 \text{ W/m}^2$ -K. The air flow enters the system at 298 K with a velocity of 1 m/s. The Reynolds number of the flow, based on the module height, is about 600. The flow is therefore treated as laminar.



#### Figure 22.1: Problem Specification

## 22.4. Setup and Solution

The following sections describe the setup and solution steps for this tutorial:

- 22.4.1. Preparation
- 22.4.2. Reading the Mesh
- 22.4.3. Manipulating the Mesh in the Viewer

22.4.4. Adding Lights
22.4.5. Creating Isosurfaces
22.4.6. Generating Contours
22.4.7. Generating Velocity Vectors
22.4.8. Creating an Animation
22.4.9. Displaying Pathlines
22.4.10. Creating a Scene With Vectors and Contours
22.4.11. Advanced Overlay of Pathlines on a Scene
22.4.12. Creating Exploded Views
22.4.13. Animating the Display of Results in Successive Streamwise Planes
22.4.15. Creating Annotation
22.4.16. Saving Picture Files
22.4.17. Generating Volume Integral Reports

### 22.4.1. Preparation

To prepare for running this tutorial:

- 1. Set up a working folder on the computer you will be using.
- 2. Go to the ANSYS Customer Portal, https://support.ansys.com/training.

#### Note

If you do not have a login, you can request one by clicking **Customer Registration** on the log in page.

- 3. Enter the name of this tutorial into the search bar.
- 4. Narrow the results by using the filter on the left side of the page.
  - a. Click ANSYS Fluent under Product.
  - b. Click **18.0** under **Version**.
- 5. Select this tutorial from the list.
- 6. Click the **postprocess\_R180.zip** link to download the input files.
- 7. Unzip the postprocess\_R180.zip file you downloaded to your working folder.

The files chip.cas.gz and chip.dat.gz can be found in the postprocess folder created after unzipping the file.

8. Use Fluent Launcher to start the **3D** version of Fluent.

Fluent Launcher displays your **Display Options** preferences from the previous session.

For more information about Fluent Launcher, see starting ANSYS Fluent using the Fluent Launcher in the Fluent Getting Started Guide.

- 9. Ensure that the **Display Mesh After Reading** and **Workbench Color Scheme** options are enabled.
- 10. Ensure that the Serial processing option is selected.
- 11. Ensure that the **Double Precision** option is disabled.

### 22.4.2. Reading the Mesh

1. Read in the case and data files chip.cas.gz and chip.dat.gz.

100001 10000 10	File $\rightarrow$	Read	→ Case	&	Data
-----------------	--------------------	------	--------	---	------

When you select the case file, Fluent will read the data file automatically.

### 22.4.3. Manipulating the Mesh in the Viewer

1. Display the mesh surfaces **board-top** and **chip**.

10001 10000 10	Setting	Up	Domain $\rightarrow$	$Mesh \rightarrow$	Display
----------------	---------	----	----------------------	--------------------	---------

💶 Mesh Display	/		<b>×</b>		
Options Nodes	Edge Type All	Surfaces Filter Text			
Edges	Feature	symmetry-19	A		
Faces	Outline	▲ Wall			
Partitions		board-ends			
Overset		board-top			
Christian I	Frankrisk Antola	bottom-wall			
Shrink Factor	Feature Angle	chip	=		
0	20	chip-bottom	-		
Outline	Interior	top-wall	-		
Adjacency		New Surface 🔻			
Display Colors Close Help					

- a. Retain the default enabling of the **Edges** option and disable the **Faces** option in the **Options** group box.
- b. Deselect all surfaces and select **board-top** and **chip** from the **Surfaces** selection list.

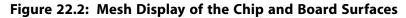
Click to deselect all surfaces. Click and select **Surface Type** under **Group By** to list the surfaces by type, as shown above.

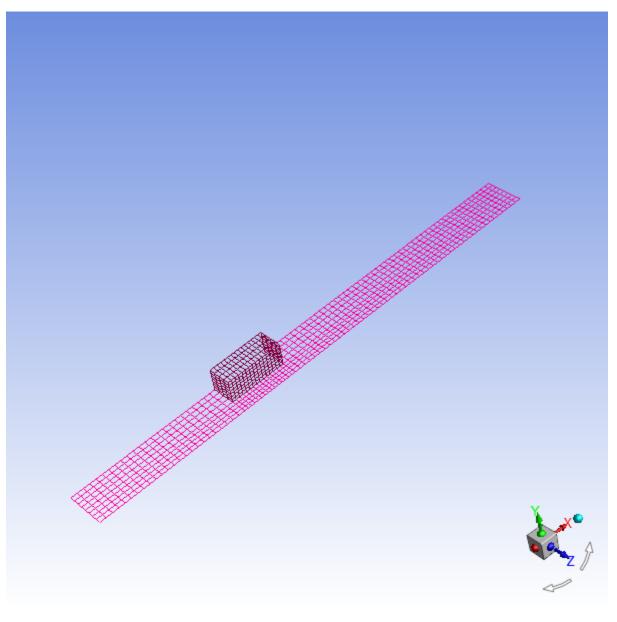
- c. Click the Colors... button to open the Mesh Colors dialog box.
  - i. Select Color by ID in the Options group box.
  - ii. Click **Reset Colors** to reset the mesh colors to the default settings and close the **Mesh Colors** dialog box.

### d. Click Display.

2. Rotate and adjust the magnification of the view.

Use the left mouse button to rotate the view. Use the middle mouse button to adjust the magnification until you obtain an enlarged display of the circuit board in the region of the chip, as shown in Figure 22.2: Mesh Display of the Chip and Board Surfaces (p. 883).





## Extra

You can click the right mouse button on one of the mesh boundaries displayed in the graphics window and its surface group, ID, and name will be displayed in the console. This feature is especially useful when you have several zones of the same type and you want to distinguish between them.

3. Display the mesh faces.

**Setting Up Domain**  $\rightarrow$  Mesh  $\rightarrow$  Display...

- a. Disable Edges and enable Faces in the Options group box.
- b. Click **Display** and close the **Mesh Display** dialog box.

## 22.4.4. Adding Lights

1. Add lighting effects.

The default light settings add a white light at the position (1,1,1). The default light is defined in the **Lights** dialog box by the **Light ID** 0 with **Direction** vectors (**X**, **Y**, **Z**) as (1, 1, 1).

 $\blacksquare Viewing \rightarrow Display \rightarrow Options...$ 

Display Options	×
Rendering	Graphics Window
Line Width 1	Active Window Close
Point Symbol (+)	1
Animation Option	Set
All	Color Scheme
Double Buffering	Workbench 🔻
Outer Face Culling	Lighting Attributes
Hidden Line Removal	Lights On
	Lighting
Hidden Surface Removal Removal Method	Gouraud 🔻
Hardware Z-buffer	Layout
	Titles
Display Timeout	Axes
Timeout in seconds 60	Ruler
	Logo
	Color
	White 🔻
	Colormap
	Colormap Alignment
	Left 🔻
Apply Info Lights	Close Help

- a. Make sure that the Lights On option is enabled in the Lighting Attributes group box.
- b. Retain **Gouraud** as the selected **Lighting** method.

*Flat* is the most basic lighting whereas *Gouraud* gives better color gradation. Note that *Gouraud* rounds off corners, and so should be used with caution on highly angular geometries.

c. Click **Apply** and close the **Display Options** dialog box.

Shading will be added to the surface mesh display (Figure 22.3: Graphics Window with Default Lighting (p. 885)).

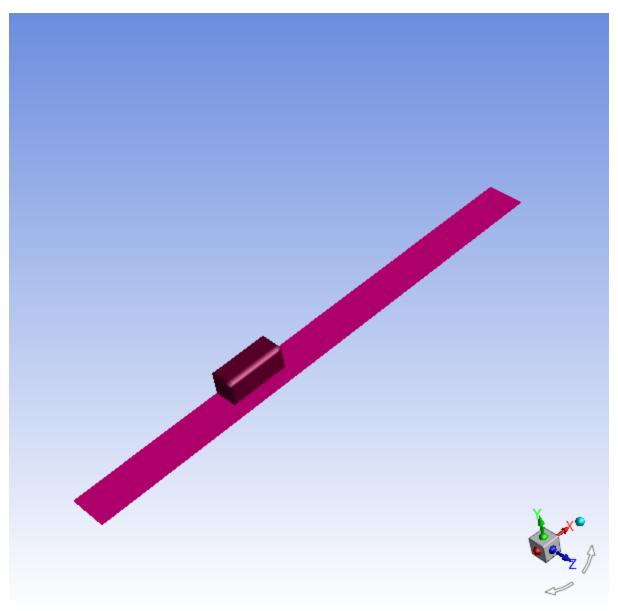


Figure 22.3: Graphics Window with Default Lighting

2. Add lights in two directions, (-1, 1, 1) and (-1, 1, -1).



💶 Lights		
Light ID 1  Light On Direction X -1  Y 1  Z 1  Use View Vector	Color Color 127  I27  Red 127  Green 127  Blue	Active Lights
	Apply Reset Close	Help

*You can also open the Lights dialog box by clicking the Lights... button in the Display Options dialog box.* 

- a. Set **Light ID** to 1.
- b. Enable Light On.
- c. Enter -1, 1, and 1 for X, Y, and Z respectively in the Direction group box.
- d. Retain the selection of Gouraud in the Lighting Method drop-down list.
- e. Make sure that the **Headlight On** option is enabled.

The **Headlight On** option provides constant lighting effect from a light source directly in front of the model, in the direction of the view. You can turn off the headlight by disabling the **Headlight On** option (the results of this action are shown in Figure 22.4: Display with Additional Lighting: - Headlight Off (p. 887)).

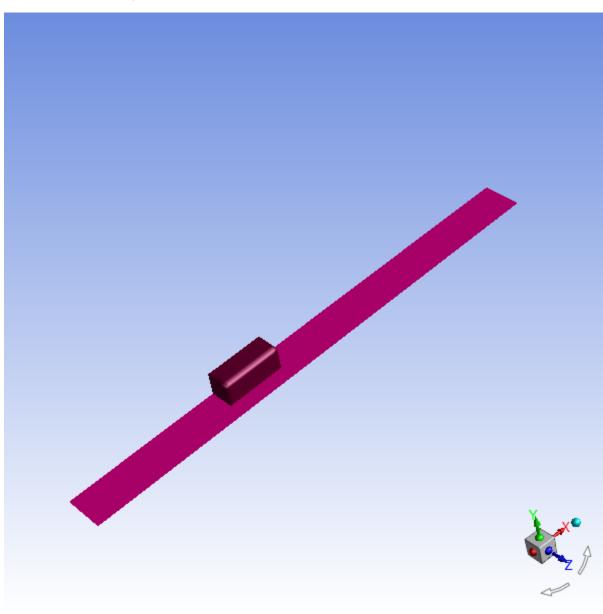
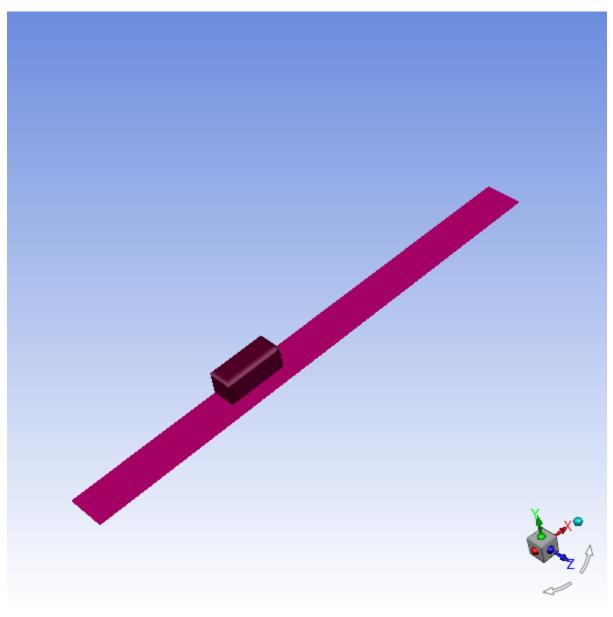


Figure 22.4: Display with Additional Lighting: - Headlight Off

## f. Click Apply.

g. Similarly, add a second light (Light ID= 2 with Light On enabled) with a Direction of (-1, 1, -1). The result will be more softly shaded display (Figure 22.5: Display with Additional Lighting (p. 888)).

Figure 22.5: Display with Additional Lighting



h. Close the Lights dialog box.

## Extra

You can use the left mouse button to rotate the ball in the **Active Lights** window to gain a perspective view on the relative locations of the lights that are currently active, and see the shading effect on the ball at the center.

You can also change the color of one or more of the lights by selecting the color from the **Color** drop-down list or by moving the **Red**, **Green**, and **Blue** sliders.

# 22.4.5. Creating Isosurfaces

To display results in a 3D model, you will need surfaces on which the data can be displayed. Fluent creates surfaces for all boundary zones automatically. Several surfaces have been renamed after reading the case file. Examples are **board-sym** and **board-ends**, which correspond to the side and end faces of the circuit board.

You can define additional surfaces for viewing the results, such as a plane in Cartesian space. In this exercise, you will create a horizontal plane cutting through the middle of the module with a Y value of 0.25 inches. You can use this surface to display the temperature and velocity fields.

- Iso-Surface х Surface of Constant = - x From Surface Filter Text Mesh... board-ends . Y-Coordinate board-sym Ξ Min (in) Max (in) board-top 0 1.1 bottom-wall Iso-Values (in) chip chip-bottom 0.25 - x From Zones Filter Text New Surface Name fluid-8 v=0.25in solid-1 solid-2 Create Compute Manage... Close Help
- 1. Create a surface of constant Y coordinate.

Postprocessing  $\rightarrow$  Surface  $\rightarrow$  Create  $\rightarrow$  Iso-Surface...

- a. Select Mesh... and Y-Coordinate from the Surface of Constant drop-down lists.
- b. Click Compute.

The **Min** and **Max** fields display the Y extents of the domain.

- c. Enter 0.25 for Iso-Values.
- d. Enter y=0.25in for New Surface Name.

#### Tip

When you are creating multiple postprocessing surfaces, it can be helpful to group

surfaces by type for viewing in lists (Click **surface Type** under **Group By**). All iso-surfaces will be grouped together.

- e. Click Create and close the Iso-Surface dialog box.
- 2. Create a clipped surface for the X coordinate of the fluid (fluid-sym).

Setting Up Domain  $\rightarrow$  Surface  $\rightarrow$  Create  $\rightarrow$  Iso-Clip...

💶 Iso-Clip			<b>—</b>
Clip to Values of Mesh	-	▼ Clip Surface fluid	× = =
X-Coordinate		🚽 fluid-sym	
	Max (in)		
1.9	3.9	New Surface Name	]
		fluid-sym-x-clip	
	Clip Compute M	lanage Close Help	h

- a. Select Mesh... and X-Coordinate from the Clip to Values of drop-down lists.
- b. Select **fluid-sym** from the **Clip Surface** selection list. You can type fluid into the **Filter Text** box to quickly find this surface.
- c. Click Compute.

The **Min** and **Max** fields display the X extents of the domain.

d. Enter 1.9 and 3.9 for Min and Max respectively.

This will isolate the area around the chip.

- e. Enter fluid-sym-x-clip for New Surface Name.
- f. Click **Clip**.
- 3. Create a clipped surface for the Y coordinate of the fluid (**fluid-sym**).

Setting Up Domain  $\rightarrow$  Surface  $\rightarrow$  Create  $\rightarrow$  Iso-Clip...

Iso-Clip		<b>•</b>
Clip to Values o Mesh	f	▼ Clip Surface fluid × 🙃 🔫
Y-Coordinate		✓ fluid-sym fluid-sym-x-clip
Min (in)	Max (in)	
0.1	0.5	New Surface Name
		fluid-sym-y-clip
	Clip Compute M	Manage Close Help

- a. Select Mesh... and Y-Coordinate from the Clip to Values of drop-down lists.
- b. Retain the selection of **fluid-sym** from the **Clip Surface** selection list.
- c. Click Compute.

The **Min** and **Max** fields display the Y extents of the domain.

d. Enter 0.1 and 0.5 for **Min** and **Max** respectively.

This will isolate the area around the chip.

- e. Enter fluid-sym-y-clip for New Surface Name.
- f. Click **Clip** and close the **Iso-Clip** dialog box.

## 22.4.6. Generating Contours

1. Display filled contours of temperature on the symmetry plane (Figure 22.6: Filled Contours of Temperature on the Symmetry Surfaces (p. 893)).

Postprocessing  $\rightarrow$  Graphics  $\rightarrow$  Contours  $\rightarrow$  New...

Contours	
Contour Name	
temperature_contou	r
Options	Contours of
Filled	Temperature
Node Values	Static Temperature 🔹
<ul> <li>Global Range</li> <li>Auto Range</li> </ul>	Min Max
Clip to Range	0 0
Draw Profiles	Surfaces Filter Text
	Symmetry
Coloring Banded	board-sym chip-sym fluid-sym
<ul> <li>Smooth</li> <li>Colormap Options</li> </ul>	symmetry-13 symmetry-19
	New Surface
(	Save/Display Compute Close Help

- a. Enter temperature\_contour for Contour Name.
- b. Ensure Filled, Node Values, Global Range, and Auto Range are enabled in the Options group box.
- c. Select Smooth for Coloring.
- d. Select Temperature... and Static Temperature from the Contours of drop-down lists.
- e. Click 📰 and select **Surface Type** under **Group By** (if surfaces are not already grouped by type).
- f. Select board-sym, chip-sym, and fluid-sym (under Symmetry in the Surfaces selection list.)
- g. Click Save/Display.
- h. Rotate and adjust the magnification of the view using the left and middle mouse buttons, respectively, to obtain the view as shown in Figure 22.6: Filled Contours of Temperature on the Symmetry Surfaces (p. 893).

## Tip

If the model disappears from the graphics window at any time, or if you are having difficulty manipulating it with the mouse, do one of the following:

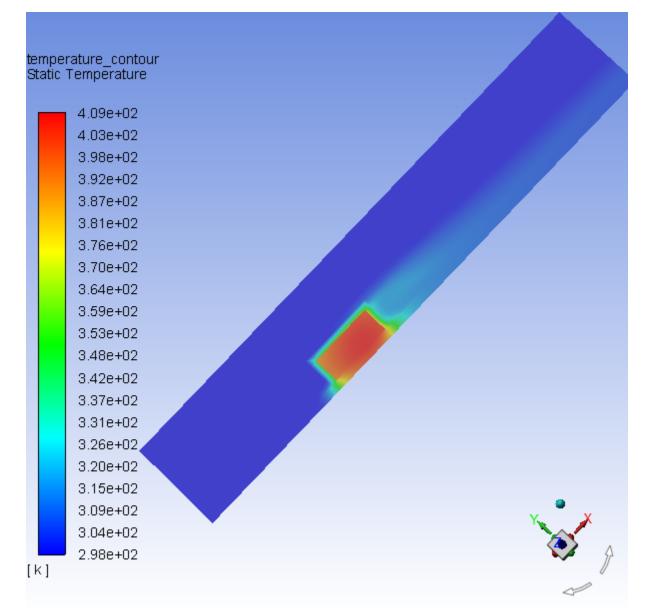
- Click the Fit to Window button in the graphics toolbar.
- Open the Views dialog box by right-clicking Graphics in the tree (under Results) and selecting Views... from the menu that opens, and then use the Default button

to reset the view. You could also click **Camera...** in this dialog box to open the **Camera Parameters** dialog box, where you could select **orthographic** from the **Projection** drop-down list to reduce the likelihood of zooming through the geometry.

• Press the **Ctrl** + **L** to revert to a previous view.

The peak temperatures in the chip appear where the heat is generated, along with the higher temperatures in the wake where the flow is recirculating.

Figure 22.6: Filled Contours of Temperature on the Symmetry Surfaces



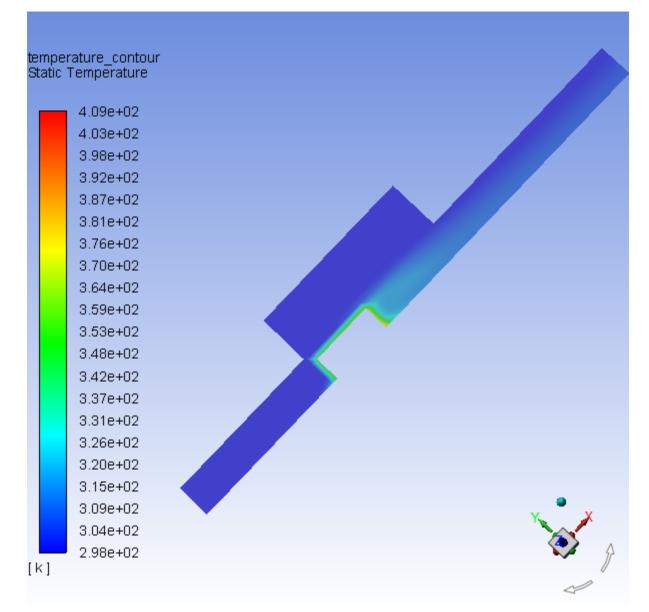
2. Display filled contours of temperature for the clipped surface (Figure 22.7: Filled Contours of Temperature on the Clipped Surface (p. 894)).

**Results**  $\rightarrow$  Graphics  $\rightarrow$  Contours  $\rightarrow$  temperature\_contour  $\stackrel{\square}{\rightarrow}$  Edit...

- a. Click **t** o deselect all surfaces from the **Surfaces** selection list and then select **fluid-sym-x-clip** and **fluid-sym-y-clip**.
- b. Click Save/Display.

A clipped surface appears, colored by temperature (Figure 22.7: Filled Contours of Temperature on the Clipped Surface (p. 894)).





3. Display filled contours of temperature on the plane, **y=0.25in** (Figure 22.8: Temperature Contours on the Surface, Y= 0.25 in. (p. 895)).



a. Click **T** to deselect all surfaces from the **Surfaces** selection list and then select **y=0.25in**.

b. Click Save/Display and close the Contours dialog box.

The filled temperature contours will be displayed on the **y=0.25in** plane.

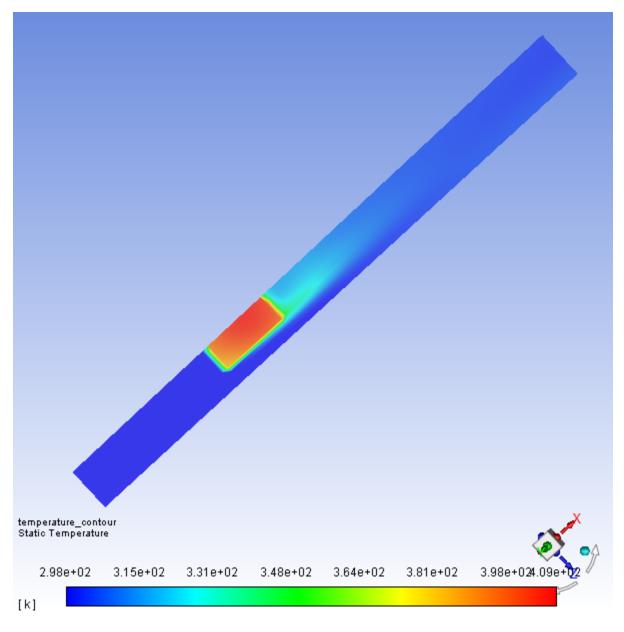
- c. Orient the view to display the contours.
- 4. Change the location of the colormap in the graphics window.

Left-click the colormap in the graphics window and drag it to the bottom of the graphics window. *This can also be accomplished using the Display Options dialog box.* 

## Tip

You can increase/decrease the size of the colormap by dragging the corners of the box that appears when you hover over the colormap.





In Figure 22.8: Temperature Contours on the Surface, Y = 0.25 in. (p. 895), the high temperatures in the wake of the module are clearly visible. You can also display other quantities such as velocity magnitude or pressure using the **Contours** dialog box.

# 22.4.7. Generating Velocity Vectors

Velocity vectors provide an excellent visualization of the flow around the module, depicting details of the wake structure.

1. Display velocity vectors on the symmetry plane through the module centerline (Figure 22.9: Velocity Vectors in the Module Symmetry Plane (p. 898)).

Vectors		×
Vector Name		
velocity_vector		
Options	Vectors of	
Global Range	Velocity	•
Auto Range	Color by	
Clip to Range	Velocity	•
Auto Scale Draw Mesh	Velocity Magnitude	•
Diaw Mesi	Min Max	
Style	0 0	
headless 🔹		
Scale Skip	Surfaces Filter Text	• )
1.9 0 🚔	Symmetry	
Vector Options	board-sym	
	chip-sym	
Custom Vectors	fluid-sym	L
Colormap Options	symmetry-13	
	symmetry-19	1
	New Surface 💌	
3	ave/Display Compute Close Help	

Postprocessing  $\rightarrow$  Graphics  $\rightarrow$  Vectors  $\rightarrow$  New...

- a. Enter velocity\_vector for Vector Name.
- b. Confirm that **Velocity** is selected under **Vectors of** and that **Color by** is set to **Velocity...** and **Velocity Magnitude**.
- c. Ensure Global Range, Auto Range, and Auto Scale are the only enabled Options.
- d. Enter 1.9 for Scale.
- e. Click **T** to deselect all surfaces from the **Surfaces** selection list and then select **fluid-sym**.
- f. Click **Save/Display** and close the **Vectors** dialog box.

g. Orient the view to display the vectors.

#### Extra

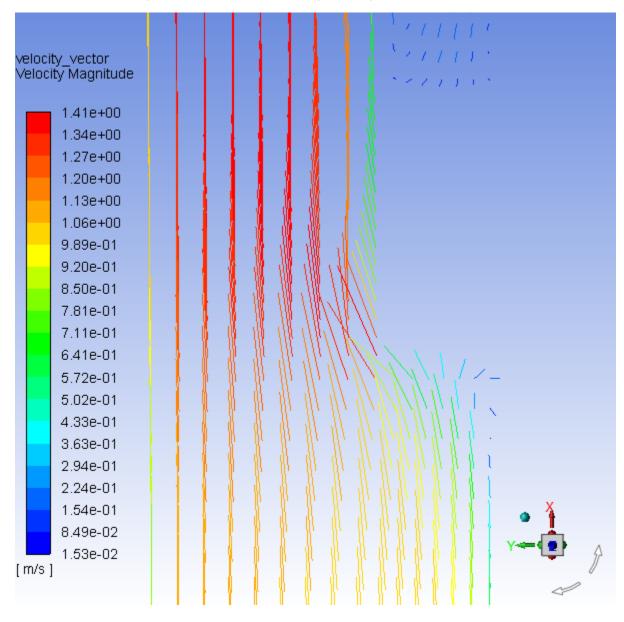
You can display velocity vectors for the clipped surfaces.



a. Deselect all surfaces from the **Surfaces** selection list and then select **fluid-sym-x-clip** and **fluid-sym-y-clip**.

b. Click Save/Display.

2. Rotate and adjust the magnification of the view to observe the vortex near the stagnation point and in the wake of the module (Figure 22.9: Velocity Vectors in the Module Symmetry Plane (p. 898)).



## Figure 22.9: Velocity Vectors in the Module Symmetry Plane

3. Plot velocity vectors in the horizontal plane intersecting the module (Figure 22.10: Velocity Vectors Intersecting the Surface (p. 900)).

Postprocessing  $\rightarrow$  Graphics  $\rightarrow$  Vectors  $\rightarrow$  Edit...

a. Enable **Draw Mesh** in the **Options** group box to open the **Mesh Display** dialog box.

🞴 Mesh Display	/		<b>—</b> ×
Options Nodes	<ul> <li>Edge Type</li> <li>All</li> </ul>	Surfaces Filter Text	
Edges	<ul> <li>Feature</li> <li>Outline</li> </ul>	symmetry-13 symmetry-19	·
Partitions     Overset		Wall board-ends	
Shrink Factor		board-top bottom-wall	
0 Outline	20 Interior	chip chip-bottom top-wall	
Adjacency		New Surface	
Display Colors Close Help			

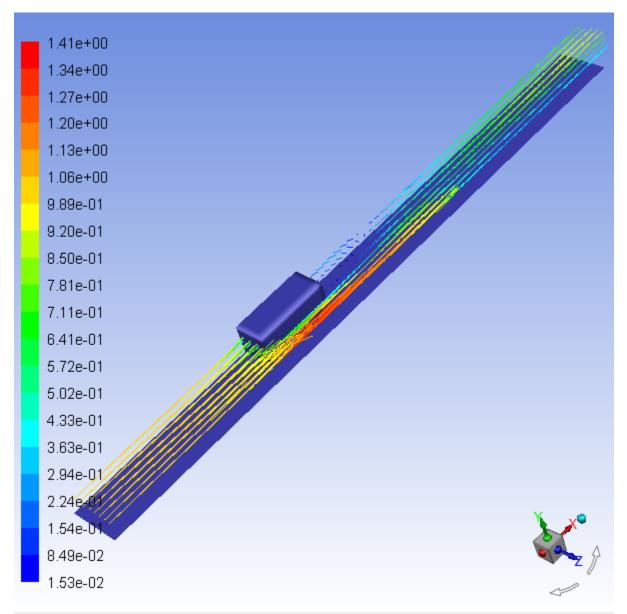
- i. Ensure that **Faces** is enabled in the **Options** group box.
- ii. Retain the selection of **board-top** and **chip** from the **Surfaces** selection list.
- iii. Click the **Colors...** button to open the **Mesh Colors** dialog box.

Mesh Colors				×
Options	Types		Colors	
<ul> <li>Color by Type</li> <li>Color by ID</li> </ul>	far-field inlet	^	green light blue	^
	interior		light gray	
Sample	outlet	Ξ	light green	
	periodic		light red light yellow	_
	rans-les-interface		magenta	=
	symmetry axis		orange	
	wall	Ŧ	pink	-
ſ	Reset Colors Close	H	elp	

- A. Select **Color by Type** in the **Options** group box.
- B. Select wall from the Types selection list.
- C. Select **light blue** from the **Colors** selection list and close the **Mesh Colors** dialog box.
- iv. Click **Display** and close the **Mesh Display** dialog box.
- b. Enter 3.8 for Scale.

- c. Deselect all surfaces.
- d. Select **y=0.25in** from the **Surfaces** selection list.
- e. Click **Display** and close the **Vectors** dialog box.
- f. Rotate the view and reduce the magnification to obtain the view as shown in Figure 22.10: Velocity Vectors Intersecting the Surface (p. 900).

Figure 22.10: Velocity Vectors Intersecting the Surface



4. Mirror the image about the chip symmetry plane (Figure 22.11: Velocity Vectors After Mirroring (p. 902)).

bottom front isometric left right top	Auto Scale Previous Save Delete Read Write	symmetry-19 symmetry-18 symmetry-7 symmetry-12 symmetry-13 Define Plane Periodic Repeats
Save Name view-0		Define

a. Select symmetry-18 from the Mirror Planes selection list.

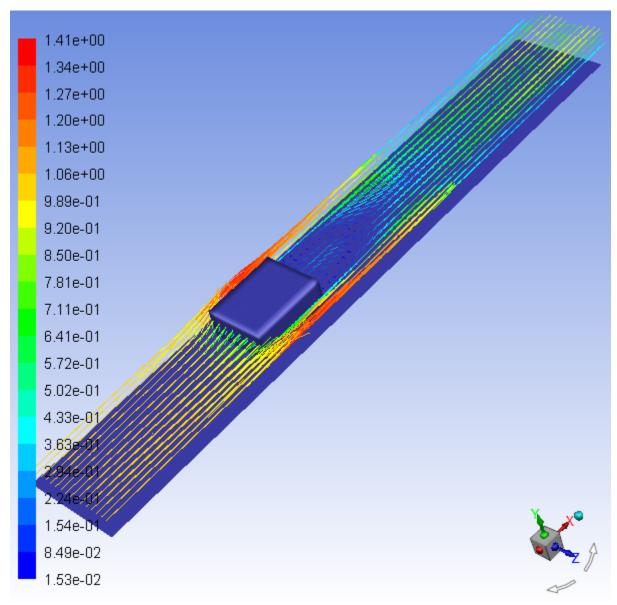
## Note

This zone is the centerline plane of the module and its selection will create a mirror of the entire display about the centerline plane.

b. Click **Apply** and close the **Views** dialog box.

The display will be updated in the graphics window (Figure 22.11: Velocity Vectors After Mirroring (p. 902)).





# 22.4.8. Creating an Animation

Using Fluent, you can animate the solution. For information on animating the solution, see Using Dynamic Meshes (p. 501). In this tutorial, you will animate between two static views of the graphics window.

You will display the surface temperature distribution on the module and the circuit board by selecting the corresponding boundaries. You will also create the key frames and view the transition between the key frames, dynamically, using the animation feature.

1. Display filled contours of surface temperature on the board-top and chip surfaces. (Figure 22.12: Filled Temperature Contours on the Chip and Board Top Surfaces (p. 904)).



Contours	
Contour Name	
temperature_contou	r
Options	Contours of
✓ Filled	Temperature
Node Values	Static Temperature 🗸 🗸
<ul> <li>Global Range</li> <li>Auto Range</li> </ul>	Min Max
Clip to Range	0 0
Draw Profiles	Surfaces Filter Text
Draw Mesh	
	▲ Wall
O la inc	board-ends
Coloring	board-top
Banded	bottom-wall
Smooth	chip 📰
	chip-bottom
Colormap Options	
	New Surface 🔻
	Save/Display Compute Close Help

- a. Ensure that **Filled** is enabled in the **Options** group box.
- b. Retain the selection of Temperature... and Static Temperature from the Contours of drop-down lists.
- c. Deselect all surfaces.
- d. Select board-top and chip from the Surfaces selection list.
- e. Click Save/Display and close the Contours dialog box.
- f. Reorient the display as needed to obtain the view shown in Figure 22.12: Filled Temperature Contours on the Chip and Board Top Surfaces (p. 904).

Figure 22.12: Filled Temperature Contours on the Chip and Board Top Surfaces (p. 904) shows the high temperatures on the downstream portions of the module and relatively localized heating of the circuit board around the module.

temperature_contour Static Temperature	
<ul> <li>4.09e+02</li> <li>4.03e+02</li> <li>3.98e+02</li> <li>3.92e+02</li> <li>3.87e+02</li> <li>3.81e+02</li> <li>3.76e+02</li> <li>3.70e+02</li> <li>3.64e+02</li> <li>3.63e+02</li> <li>3.48e+02</li> <li>3.48e+02</li> <li>3.31e+02</li> <li>3.26e+02</li> <li>3.26e+02</li> <li>3.15e+02</li> <li>3.09e+02</li> <li>3.09e+02</li> <li>3.04e+02</li> <li>2.98e+02</li> </ul>	

Figure 22.12: Filled Temperature Contours on the Chip and Board Top Surfaces

2. Create the key frames by changing the point of view.

Postprocessing  $\rightarrow$  Animation  $\rightarrow$  Scene Animation...

Animate	<b>—</b>
Playback	Key Frames
Playback Mode Play Once 🔹	Frame Keys
Start FrameIncrementEnd Frame111	1 📩 Key-1
↓ 1 Frame	Add
	Delete Delete All
Write/Record Format Key Frames	Picture Options
Write Read Close	Help

You will use the current display (Figure 22.12: Filled Temperature Contours on the Chip and Board Top Surfaces (p. 904)) as the starting view for the animation (**Frame** = 1).

a. Click **Add** in the **Key Frames** group box to create the first frame for your animation.

This will store the current display as Key-1.

- b. Magnify the view to focus on the module region.
- c. Enter 100 for Frame in the Key Frames group box.
- d. Click **Add** to create the tenth frame for your animation.

This will store the new display as **Key-100**.

I Animate			
Playback Playback Mode Play Once	Key Frames Frame Keys		
Start Frame Increment End Frame	1 - Key-1 Key-100		
Frame	Add Delete Delete All		
Write/Record Format Key Frames			
Write Read Close Help			

*The magnified view will be the one-hundredth key frame of the animation, with intermediate displays (2 through 99) to be filled in during the animation.* 

e. Rotate the view and adjust the magnification so that the downstream side of the module is in the foreground (Figure 22.13: Filled Temperature Contours on the Chip and Board Top Surfaces (p. 906)).

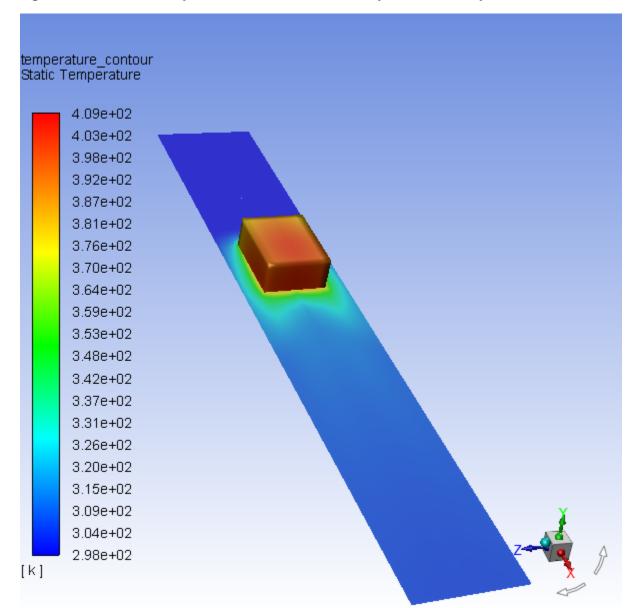


Figure 22.13: Filled Temperature Contours on the Chip and Board Top Surfaces

- f. Enter 200 for Frame.
- g. Click **Add** to create the two-hundredth frame for your animation.

This will store the new display as **Key-200**.

#### Note

You can check the display view of any of your saved key frames by selecting it in the **Keys** list.

3. View the scene animation by clicking the "play" button () in the **Playback** group box.

### Note

If your computer is having difficulty displaying the animation with 200 frames, you can increase the increment to 50.

While effective animation is best conducted on "high-end" graphics workstations, you can view scene animations on any workstation. If the graphics display speed is slow, the animation playback will take some time and will appear choppy, with the redrawing very obvious. On fast graphics workstations, the animation will appear smooth and continuous and will provide an excellent visualization of the display from a variety of spatial orientations. On many machines, you can improve the smoothness of the animation by enabling the **Double Buffering** option in the **Display Options** dialog box.

To produce a slower animation, increase the number of frames between the key frames. The more sparsely you place your key frames, the more transition frames Fluent creates between the key frames and therefore stretching out your animation.

#### Note

You can also make use of animation tools of Fluent for transient cases as demonstrated in Modeling Transient Compressible Flow (p. 267).

#### Extra

You can change the Playback Mode if you want to "auto repeat" or "auto reverse" the

animation. When you are in either of these modes, you can click the "stop" button ( to stop the continuous animation.

#### 4. Close the Animate dialog box.

## 22.4.9. Displaying Pathlines

Pathlines are the lines traveled by neutrally buoyant particles in equilibrium with the fluid motion. Pathlines are an excellent tool for visualization of complex three-dimensional flows. In this example, you will use pathlines to examine the flow around and in the wake of the module.

1. Create a rake from which the pathlines will emanate.

Postprocessing  $\rightarrow$  Surface  $\rightarrow$  Create  $\rightarrow$  Line/Rake...

Line/Rake Surface	<b>—</b>		
Options Type Line Tool Reset	Number of Points ▼ 10 ▼		
End Points			
x0 (in) 1.0	x1 (in) 1.0		
y0 (in) 0.105	y1 (in) 0.25		
z0 (in) 0.07	z1 (in) 0.07		
Select Points with Mouse			
New Surface Name			
pathline-rake			
Create Manage Close Help			

a. Select **Rake** from the **Type** drop-down list.

A rake surface consists of a specified number of points equally spaced between two specified endpoints. A line surface (the other option in the **Type** drop-down list) is a line that includes the specified endpoints and extends through the domain; data points on a line surface will not be equally spaced.

b. Retain the default value of 10 for Number of Points.

This will generate 10 pathlines.

c. Enter a starting coordinate of (1.0, 0.105, 0.07) and an ending coordinate of (1.0, 0.25, 0.07) in the **End Points** group box.

This will define a vertical line in front of the module, about halfway between the centerline and edge.

d. Enter pathline-rake for New Surface Name.

You will refer to the rake by this name when you plot the pathlines.

- e. Click Create and close the Line/Rake Surface dialog box.
- 2. Draw the pathlines (Figure 22.14: Pathlines Display (p. 910)).

Postprocessing  $\rightarrow$  Graphics  $\rightarrow$  Pathlines...  $\rightarrow$  Edit...

Pathlines			
Options Oil Flow	Style	Color by Particle Variables	
<ul> <li>Reverse</li> <li>Node Values</li> </ul>	Attributes	Particle ID	
<ul> <li>Auto Range</li> <li>Draw Mesh</li> </ul>	Step Size         (in)         Tolerance           0.001         0.001	Min Max 0 9	
<ul> <li>Accuracy Control</li> <li>Relative Pathlines</li> </ul>	Steps Path Skip 6000 🗘 0	Release from Surfaces Filter Text	
XY Plot Write to File	Path Coarsen	Iso-surface     y=0.25in	
Type	On Zone	Outlet     pressure-outlet-16	
CFD-Post   Pulse Mode  Continuous  Single	pressure-outlet-16 symmetry-12 symmetry-13 symmetry-18 symmetry-19 symmetry-7 velocity-inlet-17 wall-14 wall-15 wall-3 T	Rake-surface     pathline-rake     Symmetry     board-sym     chip-sym     fluid-sym     Highlight Surfaces     New Surface	
Display Pulse Compute Axes Curves Close Help			

- a. Enable **Draw Mesh** in the **Options** group box to open the **Mesh Display** dialog box.
  - i. Ensure that Faces is enabled in the Options group box.
  - ii. Retain the selection of **board-top** and **chip** from the **Surfaces** selection list.

These surfaces should already be selected from the earlier exercise where the mesh was displayed with velocity vectors.

- iii. Close the Mesh Display dialog box.
- b. Enter 0.001 inch for **Step Size**.
- c. Enter 6000 for Steps.

#### Note

As a general guideline to follow when you are setting these two parameters is that if you want the particles to advance through a domain of length L, the **Step Size** times the number of **Steps** should be approximately equal to L.

d. Set Path Coarsen to 5.

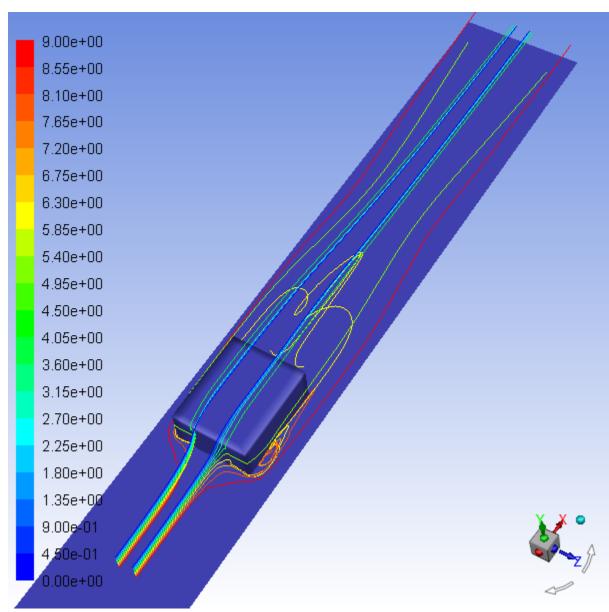
Coarsening the pathline simplifies the plot and reduces the plotting time. The coarsening factor specified for **Path Coarsen** indicates the interval at which the points are plotted for a given pathline in any cell.

e. Select pathline-rake from the Release from Surfaces selection list.

## f. Click Display.

The pathlines will be drawn on the surface.

g. Rotate and magnify the view so that the flow field is in front and the wake of the chip is visible as shown in Figure 22.14: Pathlines Display (p. 910).



## Figure 22.14: Pathlines Display

- 3. Write the pathlines to a file.
  - a. Enable Write to File in the Options group box.

The **Display** button changes to a **Write...** button.

- b. Select **Fieldview** from the **Type** drop-down list.
- c. Click the **Write...** button to open the **Select File** dialog box.

- i. Enter chip-pathline for Fieldview File.
- ii. Click **OK** to close the **Select File** dialog box.

Fluent will save the file in the Fieldview format with a .fvp extension.

4. Display pathlines as spheres.



<b>Pathlines</b>			
Options Oil Flow	Style	Color by Particle Variables	
Reverse		Particle Variables  Particle ID	
<ul> <li>Auto Range</li> <li>Draw Mesh</li> </ul>	Step Size (in) Tolerance	Min Max	
Accuracy Control     Relative Pathlines	Steps Path Skip	Release from Surfaces Filter Text	
XY Plot Write to File	Path Coarsen	Iso-surface     v=0.25in	
Type CFD-Post •	On Zone	Outlet     pressure-outlet-16	
Pulse Mode Continuous Single	pressure-outlet-16 symmetry-12 symmetry-13 symmetry-18 symmetry-19 symmetry-7 velocity-inlet-17 wall-14	<ul> <li>Rake-surface         <ul> <li>pathline-rake</li> <li>Symmetry                 board-sym                 chip-sym                 fluid-sym                 fluid-sym                      fluid-sym</li></ul></li></ul>	
	wall-15 wall-3	new Sunace ·	
Display Pulse Compute Axes Curves Close Help			

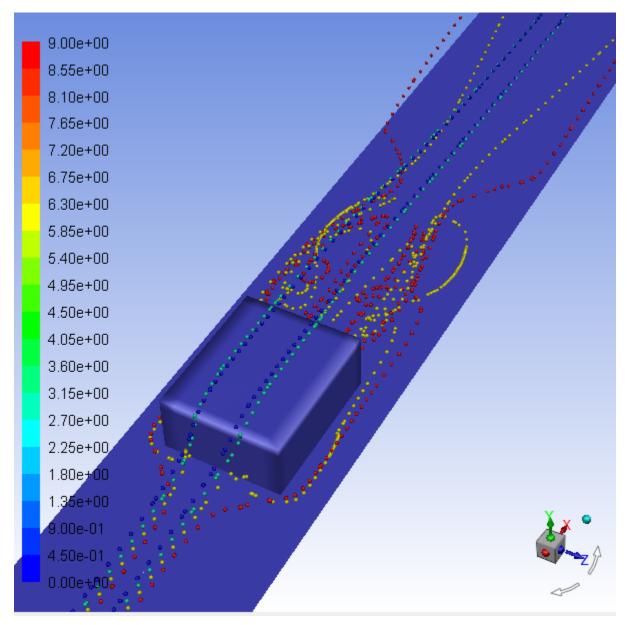
- a. Disable Write to File in the Options group box.
- b. Select **sphere** from the **Style** drop-down list.
- c. Click the Attributes... button to open the Path Style Attributes dialog box.

💶 Path Style Attribut 🔜
Diameter
0.0005
Detail
8
Scale
0
OK Cancel Help

- i. Enter 0.0005 for **Diameter**.
- ii. Click **OK** to close the **Path Style Attributes** dialog box.
- d. Enter 1 inch for **Step Size** and 1000 for **Steps** respectively.
- e. Set Path Skip to 2 and Path Coarsen to 1.
- f. Retain the selection of **pathline-rake** in the **Release from Surfaces** selection list.
- g. Click Display.

The spherical pathlines will be drawn along the surface.

## Figure 22.15: Sphere Pathlines Display



h. Select Surface ID from the lower Color by drop-down list.

i. Click **Display** and close the **Pathlines** dialog box.

This will color the pathlines by the surface they are released from (Figure 22.16: Sphere Pathlines Colored by Surface ID (p. 913)).

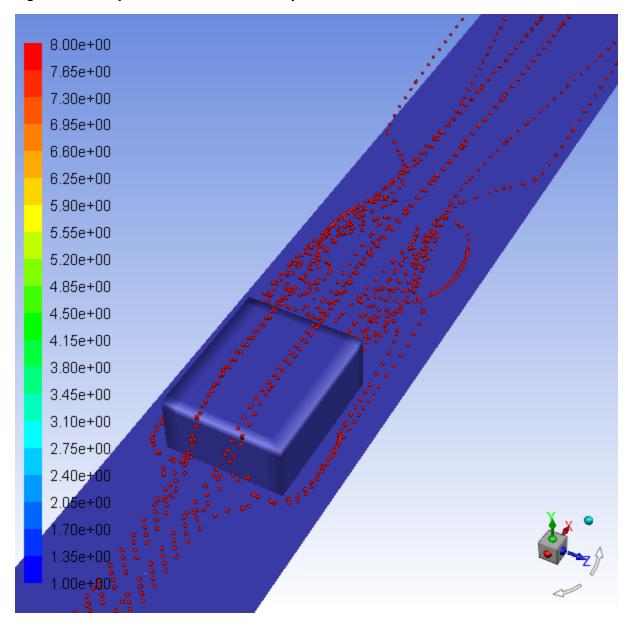


Figure 22.16: Sphere Pathlines Colored by Surface ID

### Note

As an optional exercise, you can create solution animations for pathlines using the **Animation Sequence** dialog box.

**F**Solution  $\rightarrow$  Calculation Activities  $\rightarrow$  Solution Animations  $\stackrel{\square}{\rightarrow}$  Edit...

# 22.4.10. Creating a Scene With Vectors and Contours

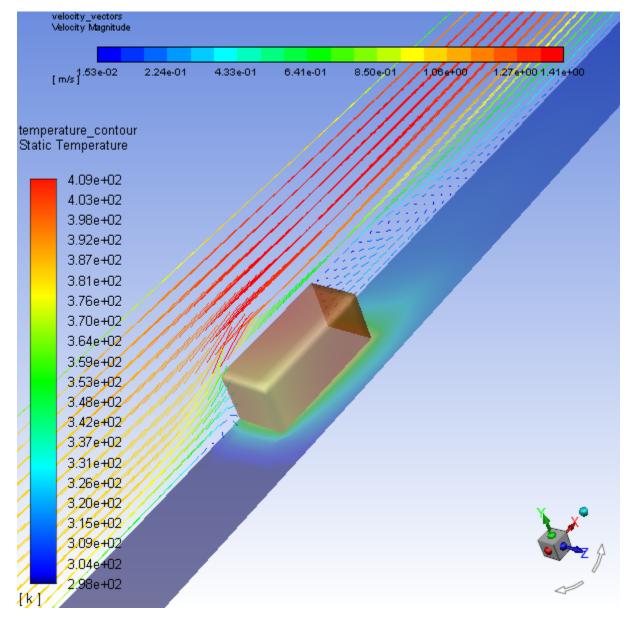
Scenes allow you to display multiple graphics plots in single window.

1. Open the **Scene** dialog box.

**Results**  $\rightarrow$  Scene  $\stackrel{\textcircled{}}{\rightarrow}$  New...

Scene	×	
Name: scene-1		
Title: scene-1		
Graphics Objects Transparency		
temperature_contour	45	
velocity-vector	0	
New Object 🔻		
Save & Display Close Help		

- 2. Enable temperature\_contour and velocity\_vector.
- 3. Increase the transparency of temperature\_contour using the Transparency slider.
- 4. Click **Save & Display** to create the scene and display it in the graphics window.
- 5. Drag the velocity vector colormap to the top of the graphics window and modify the orientation and zoom of the scene to match Figure 22.17: Temperature Contours and Velocity Vectors Scene (p. 915).



## 6. Figure 22.17: Temperature Contours and Velocity Vectors Scene

## Note

As an optional exercise, you can add a mesh graphics object to the scene as well, by selecting **Mesh...** from the **New Object** drop-down in the **Scene** dialog box. Once the mesh graphics object is saved/created, select it from the **Graphics Objects** list and click **Save & Display** in the **Scene** dialog box.

# 22.4.11. Advanced Overlay of Pathlines on a Scene

The advanced overlay capability, provided in the **Scene Description** dialog box, allows you to display more results on top of an existing display, including those not supported by the scene created in the previous section. You can demonstrate this capability by adding pathlines to the scene that you just plotted.

1. Enable the overlays feature.

$\overset{\text{\tiny Herrical Compose}}{\blacksquare} Viewing \rightarrow Graphics \rightarrow Compose$			
Scene Description		<b>—</b>	
Names [0/3] temperature_contour-12 temperature_contour-2 velocity_vector-0 Delete Geometry	Geometry Attributes Type No geometry Display Transform Iso-Value Pathlines Time Step	<ul> <li>Scene Composition</li> <li>Overlays</li> <li>Draw Frame</li> <li>Frame Options</li> </ul>	
Apply Close Help			

- a. Enable **Overlays** in the **Scene Composition** group box.
- b. Click **Apply** and close the **Scene Description** dialog box.
- 2. Add a plot of particle surface ID pathlines to the velocity vector and temperature scene.

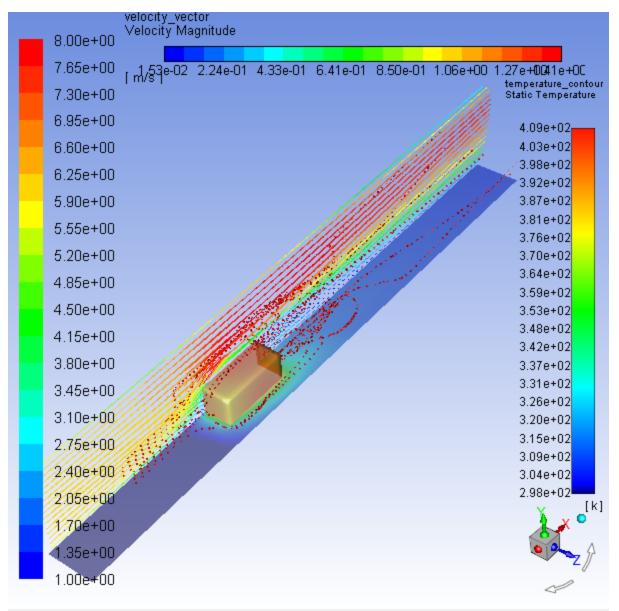
10001	Postprocessing $\rightarrow$	Graphics $\rightarrow$	Pathlines $\rightarrow$ Edit
-------	------------------------------	------------------------	------------------------------

Pathlines			
Options Oil Flow Reverse Vode Values	Style sphere Attributes Step Size (in) Tolerance	Color by Particle Variables  Surface ID	
Auto Range Draw Mesh Accuracy Control	1 0.001 Steps Path Skip	Min Max 1 Release from Surfaces Filter Text	
Relative Pathlines     XY Plot     Write to File	1000 🗘 2 🔶 Path Coarsen 1 💿	Iso-surface     y=0.25in	
Type Fieldview 👻	On Zone pressure-outlet-16	Outlet     pressure-outlet-16     Rake-surface	
Pulse Mode Continuous Single	symmetry-12 symmetry-13 symmetry-18 symmetry-19 symmetry-7 velocity-inlet-17	pathline-rake     Symmetry     Highlight Surfaces     New Surface	
Display Pulse Compute Axes Curves Close Help			

a. Disable Draw Mesh under Options.

- b. Click **Display** and close the **Pathlines** dialog box.
- c. Use the mouse to obtain the view that is shown in Figure 22.18: Overlay of Pathlines on Velocity Vectors and Temperature Contours Scene (p. 917).

Figure 22.18: Overlay of Pathlines on Velocity Vectors and Temperature Contours Scene



# 22.4.12. Creating Exploded Views

The **Scene Description** dialog box stores each display that you request and allows you to manipulate the displayed items individually. This capability can be used to generate "exploded" views, in which results are translated or rotated out of the physical domain for enhanced display. As shown in the **Scene Description** dialog box, you can experiment with this capability by displaying "side-by-side" velocity vectors and temperature contours on a streamwise plane in the module wake.

1. Clear the temperature contours, velocity vectors, and pathlines from the current display.

Click the **Close Tab** button (<sup>III</sup>) to clear the graphics window (located at on the right-hand side of the graphics window tab).

2. Create a plotting surface at X = 3 inches (named **x=3.0in**), just downstream of the trailing edge of the module.

Iso-Surface			<b>×</b>
Surface of Constant Mesh		From Surface Filter Text	
X-Coordinate Min 0 Iso-Values 3.0	Max 0	<ul> <li>Clip-surface fluid-sym-x-clip fluid-sym-y-clip</li> <li>Inlet velocity-inlet-17</li> <li>Iso-surface</li> <li>From Zones Filter Text</li> </ul>	
New Surface Name x=3.0in	Create Con	npute Manage Close Help	

## Postprocessing $\rightarrow$ Surface $\rightarrow$ Create $\rightarrow$ Iso-Surface...

## Tip

For details on creating an isosurface, see Creating Isosurfaces (p. 889).

3. Add the display of filled temperature contours on the **x=3.0in** surface.



Contours	
Options Filled	Contours of Velocity
<ul> <li>Node Values</li> <li>Global Range</li> </ul>	Velocity Magnitude
Auto Range Clip to Range	0.01532904 1.40682
Draw Profiles Draw Mesh	Surfaces Filter Text
Coloring Banded Smooth Levels Setup 20 1 +	<ul> <li>Inlet         velocity-inlet-17</li> <li>Iso-surface         x=3.0in         y=0.25in         Outlet         pressure-outlet-16         velocity-inlet-16         velocity-inlet-16</li></ul>
	Display Compute Close Help

- a. Disable Draw Mesh under Options.
- b. Deselect all surfaces in the **Surfaces** list.
- c. Select **x=3.0in** from the **Surfaces** selection list.
- d. Click **Display** and close the **Contours** dialog box.

The filled temperature contours will be displayed on the **x=3.0in** surface.

4. Add the velocity vectors on the **x=3.0in** plotting surface.

Postprocessing  $\rightarrow$  Graphics  $\rightarrow$  Vectors  $\rightarrow$  Edit...

Vectors				
Options Global Range Auto Range	Vectors of Velocity  Color by			
Clip to Range	Velocity			
Draw Mesh	Velocity Magnitude 🗸 🗸			
Style	Min Max			
headless 🔻				
Scale Skip	Surfaces Filter Text			
1.9 2	fluid-sym-y-clip			
Custom Vectors	velocity-inlet-17			
custom vectors	Iso-surface x=3.0in			
	y=0.25in			
	<ul> <li>Outlet</li> <li>pressure-outlet-16</li> </ul>			
New Surface 🔻				
	Display Compute Close Help			

- a. Enable the **Draw Mesh** option in the **Options** group box to open the **Mesh Display** dialog box.
  - i. Retain the default settings.
  - ii. Close the **Mesh Display** dialog box.
- b. Enter 1.9 for Scale.
- c. Set Skip to 2.
- d. Deselect all surfaces.
- e. Select **x=3.0in** from the **Surfaces** selection list.
- f. Click **Display** and close the **Vectors** dialog box.

The display will show the vectors superimposed on the contours of temperature at x=3.0 in.

5. Create the exploded view by translating the contour display, placing it above the vectors (Figure 22.19: Exploded Scene Display of Temperature and Velocity (p. 922)).

 $\blacksquare$  Viewing  $\rightarrow$  Graphics  $\rightarrow$  Compose...

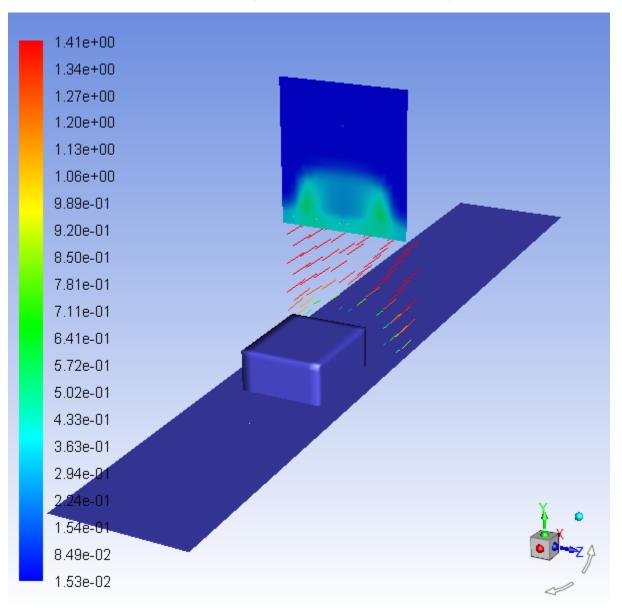
- a. Select temperature\_contour-9 from the Names selection list.
- b. Click the Transform... button to open the Transformations dialog box.

Transformatio	ns				×
Geometry Name contour-9-temperature			1eridional		
Translate	Rotate by		Rotate about	Scale	
X (in)	X (deg)	_	X (in)	X	
0	0		0	1	
Y (in)	Y (deg)		Y (in)	Y 1	
7 (-)		_			
Z (in)	Z (deg)		Z (in)	Z	_
Apply Close Help					

- i. Enter 1 inch for **Y** in the **Translate** group box.
- ii. Click Apply and close the Transformations dialog box.

The exploded view allows you to see the contours and vectors as distinct displays in the final scene (Figure 22.19: Exploded Scene Display of Temperature and Velocity (p. 922)).

- c. Deselect Overlays.
- d. Click Apply and close the Scene Description dialog box.
- e. Magnify the view, as shown in Figure 22.19: Exploded Scene Display of Temperature and Velocity (p. 922).



#### Figure 22.19: Exploded Scene Display of Temperature and Velocity

# 22.4.13. Animating the Display of Results in Successive Streamwise Planes

You may want to march through the flow domain, displaying a particular variable on successive slices of the domain. While this task could be accomplished manually, plotting each plane in turn, or using the **Scene Description** and **Animate** dialog boxes, here you will use the **Sweep Surface** dialog box to facilitate the process. To illustrate the display of results on successive slices of the domain, you will plot contours of velocity magnitude on planes along the X axis.

1. Delete the vectors and temperature contours from the display.

Click the **Close Tab** button (<sup>III</sup>) to clear the graphics window (located at on the right-hand side of the graphics window tab).

2. Display the mesh on surfaces **board-top** and **chip**.

Postprocessing  $\rightarrow$  Graphics  $\rightarrow$  Mesh  $\rightarrow$  Edit...

- 3. Use the mouse to reduce the magnification of the view in the graphics window so that the entire board surface is visible.
- 4. Generate contours of velocity magnitude and sweep them through the domain along the X axis.

Sweep Surface		<b>—</b>	
Sweep Axis X 1 Y 0 Z 0	Display Type Mesh Contours Vectors Properties	Animation Initial Value 0 Final Value 0.1651 Frames 20	
Min Value 0	Value 0.08255	Max Value 0.1651	
Create Animate Compute Close Help			

### Postprocessing $\rightarrow$ Animation $\rightarrow$ Sweep Surface...

- a. Retain the default settings in the Sweep Axis group box.
- b. Retain the default value of 0 m for **Initial Value** in the **Animation** group box.
- c. Retain 0.1651 m for **Final Value**.

#### Warning

The units for the initial and final values are in meters, regardless of the length units being used in the model. Here, the initial and final values are set to the **Min Value** and **Max Value**, to generate an animation through the entire domain.

- d. Enter 20 for Frames.
- e. Select **Contours** from the **Display Type** list to open the **Contours** dialog box.

Contours	
Options Filled	Contours of Velocity
<ul> <li>Node Values</li> <li>Global Range</li> </ul>	Velocity Magnitude
Auto Range	Min Max 0 1.405992
Draw Profiles	Surfaces Filter Text
Coloring Banded Smooth Levels Setup 20 1	fluid-sym-x-clip fluid-sym-y-clip Inlet velocity-inlet-17 Iso-surface x=3.0in y=0.25in New Surface
	OK Compute Cancel Help

- i. Disable Draw Mesh under Options.
- ii. Select Velocity... and Velocity Magnitude from the Contours of drop-down lists.
- iii. Click OK to close the Contours dialog box.
- f. Click Animate and close the Sweep Surface dialog box.

You will see the velocity contour plot displayed at 20 successive streamwise planes. Fluent will automatically interpolate the contoured data on the streamwise planes between the specified end points. Especially on high-end graphics workstations, this can be an effective way to study how a flow variable changes throughout the domain.

#### Note

You can also make use of animation tools of Fluent for transient cases as demonstrated in Modeling Transient Compressible Flow (p. 267).

### 22.4.14. Generating XY Plots

XY plotting can be used to display quantitative results of your CFD simulations. Here, you will complete the review of the module cooling simulation by plotting the temperature distribution along the top centerline of the module.

1. Define the line along which to plot results.

```
Postprocessing \rightarrow Surface \rightarrow Create \rightarrow Line/Rake...
```

Line/Rake Surface			<b>—</b>	
Options           Line Tool           Reset	Type Line		Number of Points	
End Points				
x0 (in) 2.0		x1 (in)	2.75	
y0 (in) 0.4		y1 (in)	0.4	
z0 (in) 0.01		z1 (in)	0.01	
Select Points with Mouse				
New Surface Name				
top-center-line				
Create Manage Close Help				

- a. Select Line from the Type drop-down list.
- b. Enter the coordinates of the line using a starting coordinate of (2.0,0.4,0.01) and an ending coordinate of (2.75,0.4,0.01) in the **End Points** group box.

These coordinates define the top centerline of the module.

- c. Enter top-center-line for New Surface Name.
- d. Click Create and close the Line/Rake Surface dialog box.
- 2. Plot the temperature distribution along the top centerline of the module (Figure 22.20: Temperature Along the Top Centerline of the Module (p. 928)).

Postprocessing  $\rightarrow$  Plots  $\rightarrow$  XY Plot  $\rightarrow$  New...

Solution XY Plot XY Plot Name xy-plot-1				×
Options          Options         Image: Position on X Axis         Image: Position on Y Axis         Image: Position on Y Axis         Image: Write to File         Image: Order Points		Plot Direction X 1 Y 0 Z 0	Y Axis Function Temperature Static Temperature X Axis Function Direction Vector Surfaces Filter Text x=3.0in y=0.25in ▲ Line-surface top-center-line ▲ Outlet pressure-outlet-16 ▲ Rake-surface nathline-rake New Surface ▼	4 m v xII
	Save/Plot	Axes Curv	res Close Help	н

- a. Retain the default **Plot Direction** of (1, 0, 0).
- b. Select Temperature... and Static Temperature from the Y Axis Function drop-down lists.
- c. Select top-center-line from the Surfaces selection list.

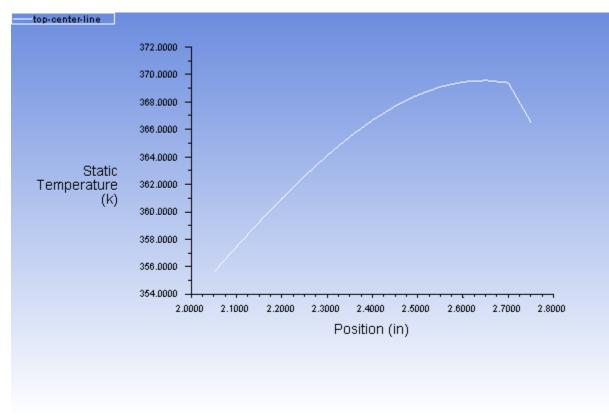
This will plot temperature vs. the X coordinate along the selected line (top-center-line).

d. Click the **Axes...** button to open the **Axes - Solution XY Plot** dialog box.

Axes - Solution XY Plot		×
Axis X Y Label	Number Format Type general Precision 3	Major Rules Color Foreground Weight 1
Options Log Auto Range Major Rules Minor Rules	Range Minimum 2.0 Maximum 2.75	Minor Rules Color dark gray Weight 1
	Apply Close H	Help

- i. Retain the selection of **X** in the **Axis** list.
- ii. Disable Auto Range in the Options group box.
- iii. Enter 2.0 for Minimum and 2.75 for Maximum in the Range group box.
- iv. Click Apply and close the Axes Solution XY Plot dialog box.
- e. Click Save/Plot and close the Solution XY Plot dialog box.

The temperature distribution (Figure 22.20: Temperature Along the Top Centerline of the Module (p. 928)) shows the temperature increase across the module surface as the thermal boundary layer develops in the cooling air flow.



#### Figure 22.20: Temperature Along the Top Centerline of the Module

#### 22.4.15. Creating Annotation

You can annotate the display with the text of your choice.

**With the set of the** 

Annotate		<b>—</b>
Names [0/0]	Annotation Text	t long the Top Centerline
	Font Specificat Name Sans Serif Color foreground	Weight       Weight       Image: Stant Size
Delete Text		
	Add Clear Close Help	

1. Enter the text describing the plot (for example, Temperature Along the Top Centerline) in the Annotation Text field.

- 2. Select 20 from the Size drop-down list in the Font Specification group box.
- 3. Click Add.

A **Working** dialog box will appear telling you to select the desired location of the text using the mouseprobe button.

4. Click the right mouse button in the graphics window where you want the text to appear, and you will see the text displayed at the selected location (Figure 22.21: A Display with Annotation (p. 929)).

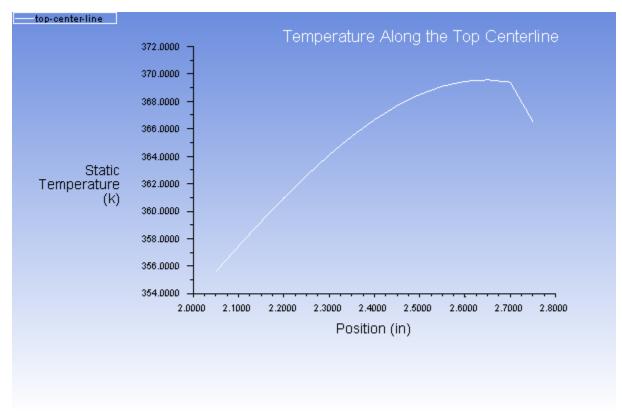
#### Extra

If you want to move the text to a new location on the screen, select the text in the **Names** selection list, click **Delete Text**, and click **Add** once again, defining a new position with the mouse.

#### Note

Depending on the size of the graphics window and the picture file format you choose, the font size of the annotation text you see on the screen may be different from the font size in a picture file of that graphics window. The annotation text font size is absolute, while the rest of the items in the graphics window are scaled to the proportions of the picture file.

Figure 22.21: A Display with Annotation



5. Close the **Annotate** dialog box.

# 22.4.16. Saving Picture Files

You can save picture files of the graphics window in many different formats, including PostScript, encapsulated PostScript, TIFF, PNG, PPM, JPEG, VRML and window dumps. Here, the procedure for saving a color PostScript file is shown.

Click the Save Picture icon-

in the toolbar to open the Save Picture dialog box.

Save Picture		<b>—</b> ×		
Format © EPS © JPEG © PPM	Coloring       File Type       Resolution			
<ul> <li>PostScript</li> <li>TIFF</li> <li>PNG</li> <li>HSF</li> </ul>	Options       Window Dump Command         Image: Command Stress       Window Dump Command         Image: White Background       Window %w			
VRML Window Dump				
Save Apply Preview Close Help				

- 1. Select **JPEG** from the **Format** list.
- 2. Retain the default selection of **Color** from the **Coloring** list.
- 3. Click the Save... button to open the Select File dialog box.
  - a. Enter a name for **Hardcopy File**.
  - b. Click **OK** to close the **Select File** dialog box.
- 4. Close the Save Picture dialog box.

## 22.4.17. Generating Volume Integral Reports

Reports of volume integrals can be used to determine the volume of a particular fluid region (that is, a fluid zone), the sum of quantities, or the maximum and minimum values of particular variables. Here we will use the volume integral reports to determine the maximum and minimum temperature in the chip, board, and the airflow.

Postprocessing  $\rightarrow$  Reports  $\rightarrow$  Volume Integrals...

<b>Volume Integrals</b>				
Report Type Mass-Average	Field Variable Temperature	•	Cell Zones Filter Text	
<ul> <li>Mass Integral</li> <li>Mass</li> </ul>	Static Temperature	•	fluid-8 solid-1	
<ul> <li>Sum</li> <li>Minimum</li> </ul>			solid-2	
<ul> <li>Maximum</li> <li>Volume</li> <li>Volume-Average</li> </ul>	Max (k) 407.439			
<ul> <li>Volume Integral</li> </ul>	Save Output Parameter			
		Compute Write Close	Нер	

- 1. Select Maximum from the Report Type list.
- 2. Select Temperature... and Static Temperature from the Field Variable drop-down lists.
- 3. Select **solid-1** from the **Cell Zones** selection list.
- 4. Click **Compute** to calculate the maximum temperature.

The maximum temperature in the solid-1 cell zone (the chip) is displayed.

5. Select **Minimum** in the **Report Type** group box and click **Compute**.

The minimum temperature in the solid-1 cell zone (the chip) is displayed.

6. Repeat the operations to determine the maximum and minimum temperatures in the **solid-2** and **fluid-8** cell zones, corresponding to the board and fluid volume, respectively.

## 22.5. Summary

This tutorial demonstrated the use of many of the extensive postprocessing features available in Fluent.

For more information on these and related features, see reporting alphanumeric data and displaying graphics in the Fluent User's Guide.

# Chapter 23: Using the Adjoint Solver – 2D Laminar Flow Past a Cylinder

This tutorial is divided into the following sections:

- 23.1. Introduction
- 23.2. Prerequisites

23.3. Problem Description

- 23.4. Setup and Solution
- 23.5. Summary

# 23.1. Introduction

ANSYS Fluent's adjoint solver is used to compute the sensitivity of quantities of interest in a fluid system with respect to the user-specified inputs, for an existing flow solution. Importantly, this also includes the sensitivity of the computed results with respect to the geometric shape of the system. The adjoint design change tool is a powerful component that can use the sensitivity information from one or more adjoint solutions to guide systematic changes that result in predictable improvements in the system performance, which can be made subject to various types of design constraints if desired.

This tutorial provides an example of how to generate sensitivity data for flow past a circular cylinder, how to postprocess the results, and how to use the data to perform a multi-objective design change that reduces drag and increases lift by morphing the mesh. The tutorial makes use of a previously computed flow solution, and demonstrates how to do the following:

- Load the adjoint solver add-on.
- Select the observable of interest.
- Access the solver controls for advancing the adjoint solution.
- Set convergence criteria and plot and print residuals.
- Advance the adjoint solver.
- · Postprocess the results to extract sensitivity data.
- Use the design change tool to modify the cylinder shape to simultaneously reduce the drag and increase the lift.

# 23.2. Prerequisites

This tutorial is written with the assumption that you have completed one or more of the introductory tutorials found in this manual:

- Introduction to Using ANSYS Fluent in ANSYS Workbench: Fluid Flow and Heat Transfer in a Mixing Elbow (p. 1)
- Parametric Analysis in ANSYS Workbench Using ANSYS Fluent (p. 73)

• Introduction to Using ANSYS Fluent: Fluid Flow and Heat Transfer in a Mixing Elbow (p. 121)

and that you are familiar with the ANSYS Fluent tree and ribbon structure. Some steps in the setup and solution procedure will not be shown explicitly.

# 23.3. Problem Description

The configuration is a circular cylinder, bounded above and below by symmetry planes. The flow is laminar and incompressible with a Reynolds number of 40, based on the cylinder diameter. At this Reynolds number, the flow is steady.

# 23.4. Setup and Solution

The following sections describe the setup steps for this tutorial:

23.4.1. Preparation
23.4.2. Step 1: Load the Adjoint Solver Add-on
23.4.3. Step 2: Define Observables
23.4.4. Step 3: Compute the Drag Sensitivity
23.4.5. Step 4: Postprocess and Export Drag Sensitivity
23.4.6. Step 5: Compute Lift Sensitivity
23.4.7. Step 6: Modify the Shape

## 23.4.1. Preparation

- 1. Set up a working folder on the computer you will be using.
- 2. Go to the ANSYS Customer Portal, https://support.ansys.com/training.

#### Note

If you do not have a login, you can request one by clicking **Customer Registration** on the log in page.

- 3. Enter the name of this tutorial into the search bar.
- 4. Narrow the results by using the filter on the left side of the page.
  - a. Click ANSYS Fluent under Product.
  - b. Click **18.0** under **Version**.
- 5. Select this tutorial from the list.
- 6. Click the **adjoint\_cylinder\_R180.zip** link to download the input files.
- 7. Unzip adjoint\_cylinder\_R180.zip.

The files cylinder\_tutorial.cas and cylinder\_tutorial.dat can be found in the adjoint\_cylinder folder created after unzipping the file.

8. Use Fluent Launcher to start the **2D** version of ANSYS Fluent, with the **Double Precision** and **Display Mesh After Reading** options enabled. For more information about starting ANSYS Fluent using the Fluent Launcher, see the Fluent Getting Started Guide, available in the help viewer or on the ANSYS Customer Portal (http://support.an-sys.com/documentation).

9. Load the converged case and data file for the cylinder geometry.

#### **File** $\rightarrow$ Read $\rightarrow$ Case & Data...

When prompted, browse to the location of the case and data files and select cylinder\_tutorial.cas to load. The corresponding data file will automatically be loaded as well.

#### Note

After you read in the mesh, it will be displayed in the embedded graphics windows, since you enabled the appropriate display option in Fluent Launcher.

The data file contains a previously computed flow solution that will serve as the starting point for the adjoint calculation. Part of the mesh and the velocity field are shown below:

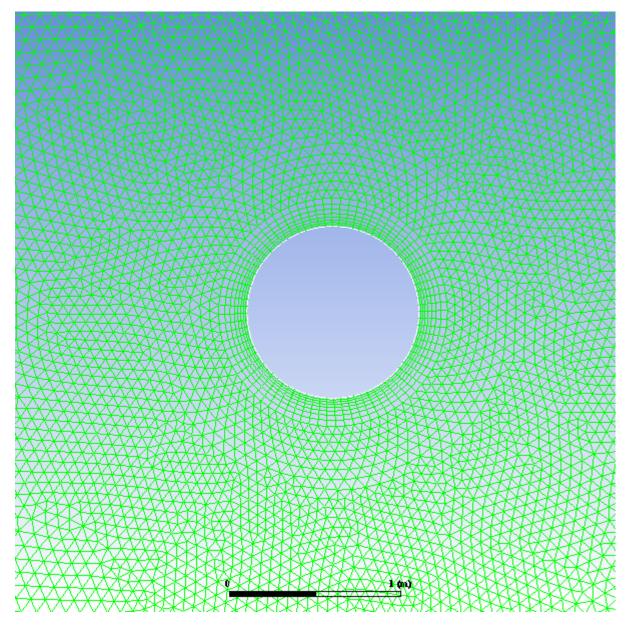


Figure 23.1: Mesh Close to the Cylinder Surface

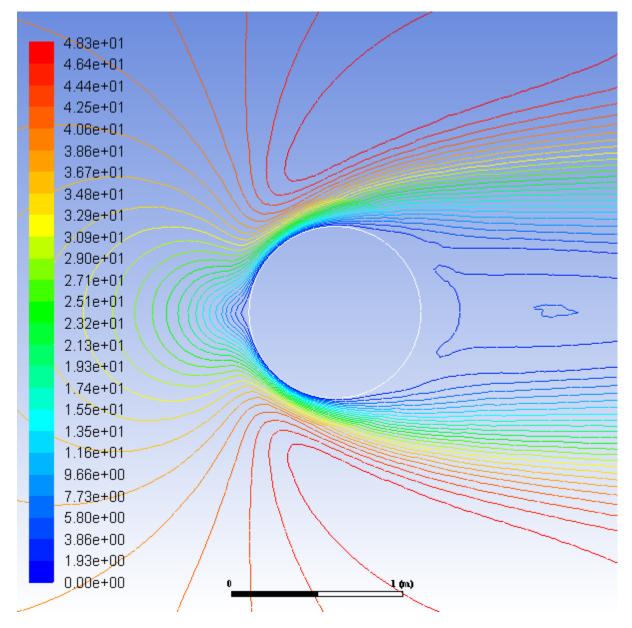


Figure 23.2: Contours of Velocity Magnitude

## 23.4.2. Step 1: Load the Adjoint Solver Add-on

The adjoint solver add-on is loaded into ANSYS Fluent by typing the following in the console:

>define/models/addon-module

#### to display the following:

Fluent Addon Modules:

- 0. None
- 1. MHD Model
- 2. Fiber Model
- 3. Fuel Cell and Electrolysis Model
- 4. SOFC Model with Unresolved Electrolyte
- 5. Population Balance Model
- 6. Adjoint Solver
- 7. Single-Potential Battery Model
- 8. Dual-Potential MSMD Battery Model
- 9. PEM Fuel Cell Model

```
10. Macroscopic Particle Model
Enter Module Number: [0] 6
```

Enter 6 to load the adjoint module. After the adjoint module is loaded, buttons become available in the **Design** tab of the ribbon within ANSYS Fluent.

## 23.4.3. Step 2: Define Observables

Begin setting up the adjoint solver by opening the **Adjoint Observables** dialog box. Here you will create lift and drag observables.

 $\blacksquare Design \rightarrow Adjoint-Based \rightarrow Observable...$ 

Figure 23.3: Adjoint Observables Dialog Box

Adjoint Observables	×
Observable Names	Evaluate
	Write
	Manage
Sensitivity Orientation	
<ul> <li>Maximize</li> <li>Minimize</li> </ul>	
Close Help	5

1. Click the Manage... button to open the Manage Adjoint Observables dialog box.

Manage Adjoint Observables				
Observables	Create			
OK Cancel Help				
in.				

Figure 23.4: Manage Adjoint Observables Dialog Box

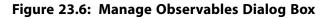
2. Click the **Create...** button in the **Manage Adjoint Observables** dialog box to open the **Create New Observable** dialog box.

Figure 23.5: Create New Observable Dialog Box

Create New Observable
<ul> <li>Observable types</li> <li>Operation types</li> </ul>
force
moment of force swirl pressure-drop fixed value surface-integral volume-integral
Name force-drag
OK Cancel Help

- 3. In the Create New Observable dialog box:
  - a. Ensure that **Observable types** is selected.

- b. Select **force** from the selection list.
- c. Enter force-drag for Name.
- d. Click **OK** to close the dialog box.
- 4. In the **Manage Adjoint Observables** dialog box, the newly created **force-drag** observable appears and must now be configured. (Figure 23.6: Manage Observables Dialog Box (p. 940)):
  - a. Select force-drag in the Observables list.



Manage Adjoint Observables	<b>—</b>	
Observables	Create	
force-drag	Apply	
	Delete	
	Rename	
Force		
	X Component	
Wall Zones [1/1] \Xi 🕎 📆	1	
wall	Y Component	
	0	
OK Cancel Help		

- b. Select wall under Wall Zones. This is the cylinder wall on which you want the force to be evaluated.
- c. Ensure that the **X-Component** direction is set to 1 and the **Y-Component** direction is set to 0.
- d. Click Apply to commit the settings for force-drag.
- 5. Repeat the process in the **Manage Adjoint Observables** dialog box to create a lift observable with the following settings:

Name	force-lift
Wall Zones	wall
X-Component	0
Y-Component	1

Tip

If the **Name** field is not available in the **Create New Observable** dialog box, select a different observable type and then select **force** again to make it available.

When you have configured the **force-lift** observable, click **OK** to commit the settings for **force-lift** and close the **Manage Adjoint Observables** dialog box.

# 23.4.4. Step 3: Compute the Drag Sensitivity

1. In the **Adjoint Observables** dialog box (Figure 23.7: Adjoint Observables Dialog Box (p. 941)) specify that you will solve for the drag sensitivity.

Figure 23.7: Adjoint Observables Dialog Box

Adjoint Observables	<b>—</b>			
Observable Names	Evaluate			
force-drag force-lift	Write			
	Manage			
<ul> <li>Sensitivity Orientation</li> <li>Maximize</li> </ul>				
Minimize				
Close Help				

a. Select force-drag in the list of Observable Names.

The selection in the **Adjoint Obervables** dialog box determines the observable for which sensitivities will be computed. You will first compute the drag sensitivities.

- b. Select **Minimize** from the **Sensitivity Orientation** list, because you are trying to reduce the drag force. This indicates that postprocessed results for the drag sensitivity will be displayed such that a reduction in drag is achieved by a design change in the positive sensitivity direction.
- c. Click **Evaluate** to print the value of the drag force on the wall in the console.

```
Observable name: force-drag
Observable Value (n) = 1271.7444
```

This value is in SI units, with n denoting Newtons.

2. Adjust the solution controls.

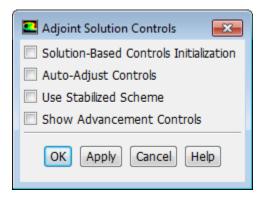
The default solution control settings are chosen to provide robust solution advancement for a wide variety of problems, including those having complex geometry, high local flow rates, and turbulence. Given sufficient iterations, a converged result can often be obtained without modifying the controls.

For this simple laminar flow case, more aggressive settings will yield faster convergence.

Open the **Adjoint Solution Controls** dialog box (Figure 23.8: Adjoint Solution Controls Dialog Box (p. 942)).

 $\blacksquare Design \rightarrow Adjoint-Based \rightarrow Solver Controls...$ 

Figure 23.8: Adjoint Solution Controls Dialog Box



a. Disable the Solution-Based Controls Initialization and Auto-Adjust Controls options.

This prevents Fluent from automatically choosing and adjusting the solution controls for you.

b. Enable Show Advancement Controls.

Adjoint Solution Controls
Solution-Based Controls Initialization
Auto-Adjust Controls
Use Stabilized Scheme
Show Advancement Controls
Advancement Controls
Apply Preconditioning
Courant Number
100
Artificial Compressibility
0.05
Flow Rate Courant Scaling
1
Under-Relaxation Factors
Adjoint Momentum
0.6
Adjoint Continuity
0.6
Adjoint Local Flow Rate
0.6
Algebraic Multigrid
Tolerance
0.1
Maximum Iterations
10
Show iterations
Defaults
OK Apply Cancel Help

c. Ensure that the **Apply Preconditioning** option is enabled.

Preconditioning can help the calculation progress in a stable manner.

d. Enter 100 for **Courant Number**.

Higher **Courant Number** values correspond to more aggressive settings / faster convergence, which is appropriate for a simple case such as this.

- e. Enter 0.05 for Artificial Compressibility.
- f. Click **OK** to close the dialog box.

3. Configure the adjoint solution monitors by opening the **Adjoint Residual Monitors** dialog box (Figure 23.9: Adjoint Residual Monitors Dialog Box (p. 944)).

**Design**  $\rightarrow$  Adjoint-Based  $\rightarrow$  Monitors...

Figure 23.9:	Adjoint	Residual	Monitors	Dialog	Box
--------------	---------	----------	----------	--------	-----

💶 Adjoint Residual Monitors			
Options Print to Console Plot	Equations		
Window 4	Residual Adjoint continuity	Check Convergence	Criteria 1e-05
Iterations to Plot 1000	Adjoint velocity Adjoint local flow rate		1e-05 0.001
OK Apply Plot Cancel Help			

In the **Adjoint Residual Monitors** dialog box, you set the adjoint equations that will be checked for convergence, as well as set the corresponding convergence criteria.

- a. Make sure that the **Print to Console** and **Plot** options are enabled.
- b. Keep the default values of 1e-05 for Adjoint continuity and Adjoint velocity, and 0.001 for Adjoint local flow rate. These settings are adequate for most cases. Make sure that the Check Convergence options are enabled.
- c. Click **OK** to close the dialog box.
- 4. Run the adjoint solver using the **Run Adjoint Calculation** dialog box (Figure 23.10: Run Adjoint Calculation Dialog Box (p. 944)).

Design  $\rightarrow$  Adjoint-Based  $\rightarrow$  Calculate...

Figure 23.10: Run Adjoint Calculation Dialog Box

💶 Run Adjoint Calcul 🔜		
Initialize		
Number of Iterations		
200 🚖		
Calculate		
Apply Close Help		

a. Click the **Initialize** button. This initializes the adjoint solution everywhere in the problem domain to zero.

- b. Set the **Number of Iterations** to 200. The adjoint solver is fully configured to start running for this problem.
- c. Click the **Calculate** button to advance the solver to convergence.

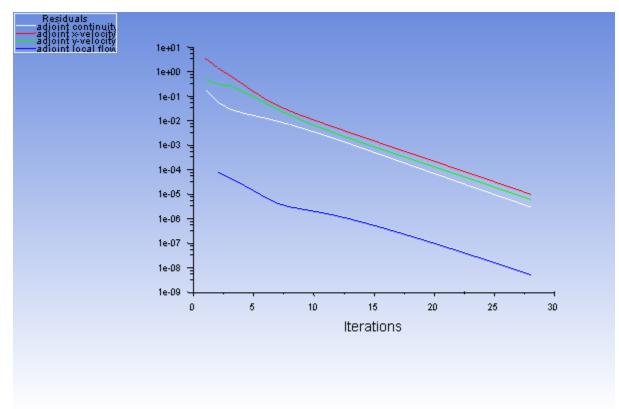


Figure 23.11: Residuals for the Converged Solution

d. When the calculation is complete, Close the Run Adjoint Calculation dialog box.

## 23.4.5. Step 4: Postprocess and Export Drag Sensitivity

In this section, postprocessing options for the adjoint solution are presented.

### 23.4.5.1. Boundary Condition Sensitivity

1. Open the Adjoint Reporting dialog box (Figure 23.12: Adjoint Reporting Dialog Box (p. 946)).

**Design**  $\rightarrow$  Adjoint-Based  $\rightarrow$  Reporting...

Figure 23.12: Adjoint Reporting Dialog Box

Adjoint Reporting
Boundary choice
inlet pressure_outlet.6 wall
Report Write Close Help

2. Select **inlet** under **Boundary choice** and click the **Report** button to display a report in the console of the available scalar sensitivity data on the inlet:

```
Updating shape sensitivity data.
Done.
Boundary condition sensitivity report: inlet
Observable: force-drag
Velocity Magnitude (m/s): 40 Sensitivity ((n)/(m/s)): 54.553894
Decrease Velocity Magnitude to decrease force-drag
```

3. Close the Adjoint Reporting dialog box.

#### 23.4.5.2. Momentum Source Sensitivity

1. Open the **Contours** dialog box.

Postprocessing  $\rightarrow$  Graphics  $\rightarrow$  Contours  $\rightarrow$  Edit...

2. Select Sensitivities... and Sensitivity to Body Force X-Component (Cell Values) from the Contours of drop-down lists.

Contours	
Options Filled	Contours of Sensitivities
✓ Node Values ✓ Global Range	Sensitivity to Body Force X-Component (Cell Values)
Auto Range Clip to Range	-1.070226 0.1695402
Draw Profiles Draw Mesh	Surfaces Filter Text
Coloring Banded Smooth Levels Setup 25 1	bottom default-interior:008 inlet pressure_outlet.6 top wall
	New Surface   Display Compute Close Help

Figure 23.13: Contours Dialog Box When Plotting Adjoint Fields

- 3. Enter 25 for Levels.
- 4. Click **Display** to view the contours (Figure 23.14: Adjoint Sensitivity to Body Force X-Component Contours (p. 948)) and then **Close** the **Contours** dialog box.

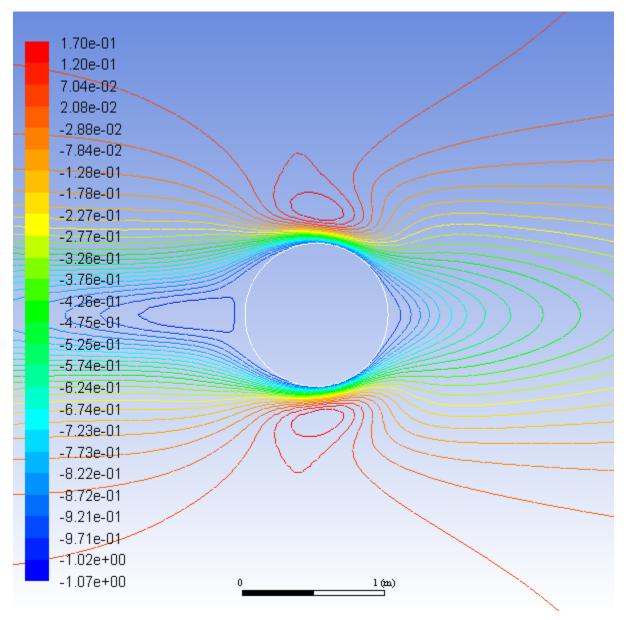


Figure 23.14: Adjoint Sensitivity to Body Force X-Component Contours

Figure 23.14: Adjoint Sensitivity to Body Force X-Component Contours (p. 948) shows how sensitive the drag on the cylinder is to the application of a body force in the *X*-direction in the flow. If a body force is applied directly upstream of the cylinder, for example, the disturbed flow is incident on the cylinder and modifies the force that it experiences.

The contours also show how the computed drag is affected by lack of convergence in the original X-momentum equation for the flow.

## 23.4.5.3. Shape Sensitivity

1. Open the Vectors dialog box (Figure 23.15: Vectors Dialog Box (p. 949))

```
Postprocessing \rightarrow Graphics \rightarrow Vectors \rightarrow Edit...
```

Figure 23.15	: Vectors	Dialog Box
--------------	-----------	------------

<b>U</b> vectors					
Options Ø Global Range	Vectors of Sensitivity to Shape	1			
Auto Range Clip to Range	Color by Sensitivities	ן			
<ul> <li>Auto Scale</li> <li>Draw Mesh</li> </ul>	Sensitivity to Mass Sources (Cell Values)				
Style	Min Max -23.88581 29.30987				
Scale Skip	Surfaces Filter Text	]			
Vector Options	default-interior:008 inlet				
Custom Vectors	pressure_outlet.6 top				
	wall New Surface				
	Display Compute Close Help	-			

- 2. Select **Sensitivity to Shape** from the **Vectors of** drop-down list.
- 3. Select **Sensitivities...** and **Sensitivity to Mass Sources (Cell Values)** from the **Color by** drop-down lists.
- 4. Select wall from the Surfaces selection list.
- 5. Click the **Display** button to view the vectors (Figure 23.16: Shape Sensitivity Colored by Sensitivity to Mass Sources (Cell Values) (p. 950)) and then **Close** the **Vectors** dialog box.

2.93 $e$ +01 2.72 $e$ +01 2.51 $e$ +01 2.29 $e$ +01 2.08 $e$ +01 1.87 $e$ +01 1.65 $e$ +01 1.44 $e$ +01 1.23 $e$ +01 1.02 $e$ +01 8.03 $e$ +00 5.90 $e$ +00 3.77 $e$ +00 1.65 $e$ +00 -4.81 $e$ -01 -2.61 $e$ +00 -4.74 $e$ +00 -8.99 $e$ +00 -1.11 $e$ +01 -1.32 $e$ +01	
-6.86e+00 -8.99e+00	
-1.54e+01	
-1.75e+01	
-1.96e+01	
-2.18e+01	
-2.39e+01	01(m)

Figure 23.16: Shape Sensitivity Colored by Sensitivity to Mass Sources (Cell Values)

This plot shows how sensitive the drag on the cylinder is to changes in the surface shape. The drag is affected more significantly if the cylinder is deformed on the upstream rather than the downstream side. Maximum effect is achieved by narrowing the cylinder in the cross-stream direction.

### 23.4.5.4. Exporting Drag Sensitivity Data

Before computing the sensitivity for the force-lift observable, you need to define the region that will be subject to geometry morphing, and export the drag sensitivity data so it can be used later in the multi-objective optimization.

- 1. Open the **Design Tool** dialog box.
  - **Design**  $\rightarrow$  Adjoint-Based  $\rightarrow$  Design Tool...

Design Change	Objectives	Region	Region Conditions	Design Conditions	Numerics	
ones To Be Modified [0/5]		led Conditions [0/0]		rne vg 🔘 Objective reference cha	nga	
nlet netuure_cutlet.6 pp val			Parameters	Freeform S	cale Factor	0.1
ommands Check	Results	Fie Name	Observable	Value	Weight	Expected change
	Results	File Name force drag	Observable force-drag	Value 1271.744	Weight	Expected change
Check	Results					
Check Strict Conditions Calculate Design Change Write Expected Changes Export Displacements Modify Nesh Export STL	Results					

Figure 23.17: The Design Tool Dialog Box

2. In the **Region** tab, define the region that will be modified for the design change.

Region Geometry Cartesian 💌	
Cartesian Region Get Bounds	Box Smaller Box
Show Bounding Box	
X Min (m)	X Max (m)
-1	1
Y Min (m)	Y Max (m)
-1	1
Update Region Expor	t Sensitivities

- a. Ensure that **Cartesian** is selected from the **Region Geometry** drop-down list.
- b. Click **Get Bounds...**.
- c. Select wall in the Bounding Box Definition dialog box and click OK.

This will initialize the morphing region to the bounding box around the cylinder **wall**.

d. Click **Update Region** to update the view of the bounding box illustration in the graphics window.

You can use the **Mesh Display** dialog box to also display the mesh, in order to review it prior to morphing.

e. Click **Larger Box** several times until the X and Y Limits are ±1.907349 m (Figure 23.18: Morphing Region Around Cylinder (p. 952)).

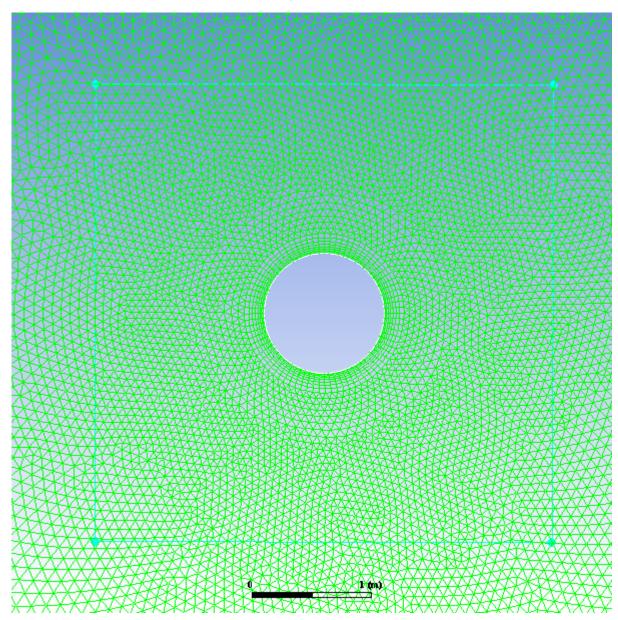


Figure 23.18: Morphing Region Around Cylinder

f. Click **Export Sensitivities...** and save the sensitivity data as force-drag.s.

## 23.4.6. Step 5: Compute Lift Sensitivity

1. Select **force-lift** from the **Observable Names** selection list and select **Maximize** from the **Sensitivity Orientation** list in the **Adjoint Observables** dialog box.

 $\blacksquare Design \rightarrow Adjoint-Based \rightarrow Observable...$ 

Adjoint Observables	<b>—</b>
Observable Names	Evaluate
force-drag	Write
force-lift	Manage
Sensitivity Orientation	
Maximize	
Minimize	
Close Hel	P 

2. Initialize and Calculate the adjoint solution using the Run Adjoint Calculation dialog box to obtain the sensitivities for the **force-lift** observable.

**Design**  $\rightarrow$  Adjoint-Based  $\rightarrow$  Calculate...

You can export the sensitivity data for the lift observable as you did for the drag, but it is not strictly necessary if you plan to perform the multi-objective optimization in the current Fluent session.

## 23.4.7. Step 6: Modify the Shape

In this section, you will load the previously saved **force-drag** sensitivity data and perform the multiobjective design change.

1. Open the **Design Tool** dialog box if it is not already open.

```
Design \rightarrow Adjoint-Based \rightarrow Design Tool...
```

**force-lift** is now displayed in the **Design Change** tab because it is the currently selected observable. The **Design Change** tab functions as a dashboard for the design modification, where you can select which boundaries are subject to modification, enable or disable conditions that you have defined, specify relative weighting if you have multiple freeform objectives, and view predicted results. You will return to it to perform the design change after you have configured the objectives and the morphing region.

- 2. Load the previously saved **force-drag** sensitivity data.
  - a. Open the **Objectives** tab.

The force-lift observable is already listed because Include current data is enabled.

Objectives	
Include current data	
	1
force-lift	
Manage Data Apply	

- b. Click Manage Data... to open the Manage Sensitivity Data dialog box.
- c. Click Import Sensitivities... and select the force-drag.s file you created earlier. Click OK.
- d. Close the Manage Sensitivity Data dialog box.
- 3. Define the objective for each observable.

For this example, you will seek a design change that increases the lift and results in a 10% reduction in drag.

- a. In the **Objectives** tab, select the **force-lift** observable. The current value of the lift is displayed along with options to specify the objective for the lift.
- b. Select Increase Value from the Objective list.

This indicates that you want to increase the lift, but are not prescribing a specific target change.

c. Enter 100 for Target/Reference Change.

This setting is used to normalize the scale of the change in value of the observable, which can be important in cases where multiple observables are considered that may be of different scales.

### d. Click Apply.

Include current data	
	Observable
force-lift	force-lift
force-drag.s	Value
-	0.4879272
	Objective
	Increase Value
	Decrease Value
	Target Change In Value
	None
	Target/Reference Change
	100
	As Percentage
Manage Data Apply	

- e. Select **force\_drag.s** in the list of observables.
- f. Select Target Change In Value from the Objective list.

This indicates that you are prescribing a specific change in the value of the observable, rather than a freeform increase or decrease.

g. Enter -10 for Target/Reference Change and enable the As Percentage option.

10% is a generally a reasonable maximum target change for a design change. Using a target change that is too large may result in very large deformations and/or overshooting the local op-timum.

h. Click **Apply**.

4. Configure the morphing region.

You already specified the dimensions of the region earlier when exporting the **force-drag** sensitivity. Now you will also configure the control-point density.

- a. Click the **Region Conditions** tab in the **Design Tool** dialog box.
- b. Enter 30 for **Points** in the **X Motion** and **Y Motion** group boxes.
- c. Click Apply.

Many other settings are available in the **Region Conditions** tab, including constraints on controlpoint motion, symmetry conditions, and continuity conditions. For additional information, see the section on defining region conditions in the Fluent Advanced Add-On Modules manual, available in the help viewer or on the ANSYS Customer Portal (http://support.ansys.com/documentation).

- 5. Compute the design change and modify the mesh.
  - a. Return to the **Design Change** tab.
  - b. Select wall in the Zones To Be Modified selection list.

Only zones that are selected in the **Zones To Be Modified** list (or that have prescribed motions applied) will be modified as part of the design change.

- c. If multiple freeform objectives were defined (that is, multiple objectives with **Increase Value** or **Decrease Value** selected in the **Objectives** tab), you would need to specify the **Weight** for each. In this case only one objective (**force-lift**) is freeform, so no input is required for **Weight**.
- d. Retain the default settings of **Control-point spacing** for **Freeform Scaling Scheme**, and **0.1** for **Freeform Scale Factor**.

These settings allow you to adjust the magnitude of the attempted design change (**Freeform Scale Factor**) and the basis for the scaling (**Freeform Scaling Scheme**).

e. Click Calculate Design Change.

The **Results** list is updated to reflect the **Expected change** for each observable.

File Name	Observable	Value	Weight	Expected change
force-lift	force-lft	0.4879272	1	127.5843
force-drag.s	force-drag	1271.744	1	-127.1744

Note that the drag is predicted to decrease by 10% as you requested, and the lift is predicted to increase.

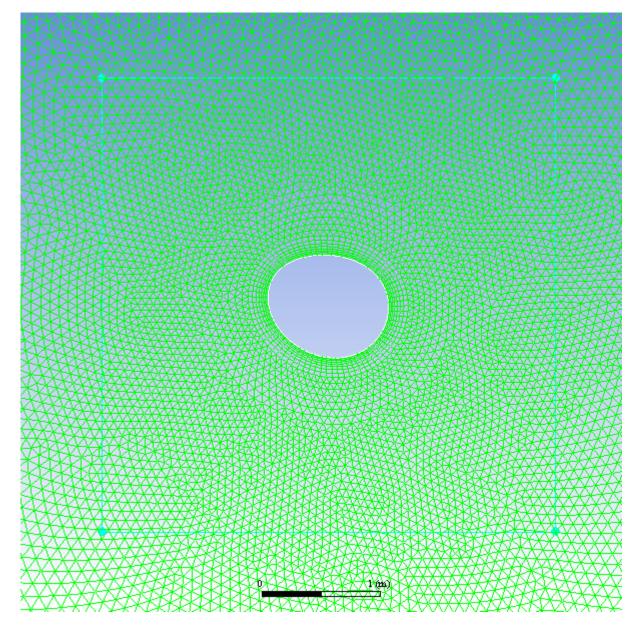
f. Click the **Modify Mesh** button to apply the calculated mesh deformation that will reposition the boundary and interior nodes of the mesh. Information regarding the mesh modification is printed in the console:

```
Updating mesh (steady, mesh iteration = 00001, pseudo time step 1.0000e+00)...
Dynamic Mesh Statistics:
Minimum Volume = 3.46260e-04
Maximum Volume = 6.36270e-01
Maximum Cell Skew = 3.69343e-01 (cell zone 11)
Minimum Orthogonal Quality = 6.30657e-01 (cell zone 11)
```

g. Display the new mesh geometry.

Postprocessing  $\rightarrow$  Graphics  $\rightarrow$  Mesh  $\rightarrow$  Edit...

The effect on the mesh is shown in Figure 23.19: Mesh After Deformation (p. 958):



### Figure 23.19: Mesh After Deformation

h. Re-converge the conventional flow calculation for this new geometry in the **Run Calculation** task page.



The currently loaded case file already has report definitions defined for lift and drag, or you can **Evaluate** the new values in the **Adjoint Observables** dialog box.

**Design**  $\rightarrow$  Adjoint-Based  $\rightarrow$  Observable...

The new values for drag and lift are reported to be:

```
Observable name: force-drag
Observable Value (n): 1151.3445
```

```
Observable name: force-lift
Observable Value (n): 122.92812
```

Note that the drag has changed by -120.4 N or -9.5% compared to the drag on the undeformed cylinder. This value compares very well with the change of -127.2 N (-10%) that was predicted from the adjoint solver. The lift has increased by 122.4 N, which again compares very well with the predicted change of 127.6 N.

## 23.5. Summary

This tutorial has demonstrated how to use the adjoint solver add-on to compute the sensitivity of the drag and lift on a circular cylinder to various inputs for a previously computed flow field. The process of setting up and running the adjoint solver was illustrated. The steps to perform various forms of postprocessing were also described. The design change tool was used to make a multi-objective change to the design that reduced the drag and increased the lift in a predictable manner.

This example considered multiple objectives at a single flow condition. Another powerful application of the design tool is to perform multi-objective design changes using sensitivities computed for multiple flow conditions. This allows you to identify design changes that improve performance across a range of anticipated operating conditions, potentially of differing importance. The design tool also offers a rich set of additional capabilities for including prescribed deformations, bounding planes / surfaces, and fixed-wall constraints in your multi-objective design change. For full details about how to use the design tool, refer to the Fluent Advanced Add-On Modules documentation, available in the help viewer or on the ANSYS Customer Portal (http://support.ansys.com/documentation).

# Chapter 24: Simulating a Single Battery Cell Using the MSMD Battery Model

This tutorial is divided into the following sections: 24.1. Introduction 24.2. Prerequisites 24.3. Problem Description 24.4. Setup and Solution 24.5. Summary 24.6. Appendix 24.7. References

# 24.1. Introduction

This tutorial is used to show how to set up a battery cell simulation in ANSYS Fluent.

This tutorial demonstrates how to do the following:

- Load the Battery module add-on
- Set up a battery cell simulation using the NTGK battery submodel
- Perform the calculations for different battery discharge rates and compare the results using the postprocessing capabilities of ANSYS Fluent
- Use the reduced order method (ROM) in a battery simulation
- Simulate a battery pulse discharge
- Introduce external and internal short-circuits in a battery simulation

# 24.2. Prerequisites

This tutorial is written with the assumption that you have completed one or more of the introductory tutorials found in this manual:

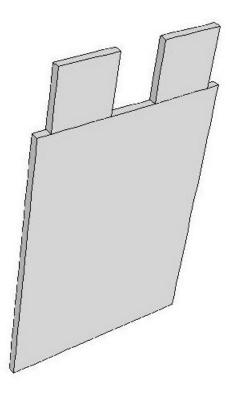
- Introduction to Using ANSYS Fluent in ANSYS Workbench: Fluid Flow and Heat Transfer in a Mixing Elbow (p. 1)
- Parametric Analysis in ANSYS Workbench Using ANSYS Fluent (p. 73)
- Introduction to Using ANSYS Fluent: Fluid Flow and Heat Transfer in a Mixing Elbow (p. 121)

and that you are familiar with the ANSYS Fluent tree and ribbon structure. Some steps in the setup and solution procedure will not be shown explicitly.

# 24.3. Problem Description

The discharge behavior of a lithium-ion battery described in the Kim's paper [2] will be modeled in this tutorial. You will use the NTGK model. The battery is a 14.6 Ah LiMn2O4 cathode/graphite anode battery. The geometry of the battery cell is shown in Figure 24.1: Schematic of the Battery Cell Problem (p. 962). You will study the battery's behavior at different discharge rates.

## Figure 24.1: Schematic of the Battery Cell Problem



For external and internal short-circuit treatment, you will consider an extreme case where external and internal short-circuits occur at the same time. You will simulate post-short-circuit battery processes. You can assume that the internal short is caused by a nail penetration occurring near the center of the battery.

# 24.4. Setup and Solution

The following sections describe the setup and solution steps for this tutorial:

24.4.1. Preparation

- 24.4.2. Reading and Scaling the Mesh
- 24.4.3. Loading the MSMD battery Add-on
- 24.4.4. NTGK Battery Model Setup

24.4.5. Postprocessing

24.4.6. Simulating the Battery Pulse Discharge Using the ECM Model

24.4.7. Using the Reduced Order Method (ROM)

24.4.8. External and Internal Short-Circuit Treatment

## 24.4.1. Preparation

1. Set up a working folder on the computer you will be using.

2. Go to the ANSYS Customer Portal, https://support.ansys.com/training.

## Note

If you do not have a login, you can request one by clicking **Customer Registration** on the log in page.

- 3. Enter the name of this tutorial into the search bar.
- 4. Narrow the results by using the filter on the left side of the page.
  - a. Click ANSYS Fluent under Product.
  - b. Click **18.0** under **Version**.
- 5. Select this tutorial from the list.
- 6. Click the **battery\_cell\_R180.zip** link to download the input files.
- 7. Unzip battery\_cell\_R180.zip.

The input file unit\_battery.msh, unit\_battery.cas, and unit\_battery.dat can be found in the battery\_cell folder created after unzipping the file.

- 8. Use Fluent Launcher to start the **3D** version of ANSYS Fluent.
- 9. Enable **Double Precision**.
- 10. Ensure that the **Display Mesh After Reading** and **Workbench Color Scheme** options are enabled.
- 11. Select either **Serial** or **Parallel** under **Processing Options**. In this tutorial, **Serial** is selected for demonstration only.

# 24.4.2. Reading and Scaling the Mesh

1. Read the mesh file unit\_battery.msh.

File → Read → Mesh...

When prompted, browse to the location of the unit\_battery.msh and select the file.

Once you read in the mesh, it is displayed in the embedded graphics windows.

The geometry is already in the correct scale. You don't need to scale it.

2. Check the mesh.

**Setting Up Domain**  $\rightarrow$  Mesh  $\rightarrow$  Check

# 24.4.3. Loading the MSMD battery Add-on

1. To load the MSMD battery add-on into ANSYS Fluent, type the following in the console window:

define/models/addon-module

### A list of ANSYS Fluent add-on modules is displayed.

Fluent Addon Modules:

- Note and the following of the second s
- 2. Enter 8 to load the MSMD battery itself.

During the loading process, a scheme library containing the graphical and text user interface, and a library of user-defined functions (UDFs) containing a set of UDFs for the battery module are loaded into ANSYS Fluent. Fluent reports the progress in the console.

Once the MSMD battery add-on is loaded, **MSMD Battery Model** appears in the **Models** task page and under the **Models** tree branch. The UDF library also becomes visible as a new entry in the UDF Library Manager dialog box.

## 24.4.4. NTGK Battery Model Setup

The following sections describe the setup steps for this tutorial: 24.4.4.1. Specifying Solver and Models

24.4.4.2. Defining New Materials for Cell and Tabs

24.4.4.3. Defining Cell Zone Conditions

24.4.4. Defining Boundary Conditions

24.4.4.5. Specifying Solution Settings

24.4.4.6. Obtaining Solution

## 24.4.4.1. Specifying Solver and Models

1. In the **Solver** group of the **Setting Up Physics** ribbon tab, enable a time-dependent calculation.

Setting Up Physics → Solver → Transient

2. To solve for the temperature field, enable the Energy equation (in the Models group).

Setting Up Physics → Models → Energy

3. Enable the battery model.

Setting Up Physics → Models → More → Dual Potential MSMD Battery Model

a. In the MSMD Battery Model dialog box, select Enable MSMD Battery Model.

The dialog box expands to display the battery model's settings.

### Figure 24.2: Model Options

SMD Battery Model			<b>—</b>	
Enable MSMD Battery Model				
Model Options Model Paramet	ers Conductive Zones	Electric Contacts	Advanced Option	
E-Chemistry Models <ul> <li>NTGK Emperical Model</li> <li>Equivalent Circuit Model</li> <li>Newman P2D Model</li> <li>User-defined E-Model</li> </ul>	Solution Method for E-Fie Solving Transport Equ Reduced Order Method	ation		
Energy Source Options	Solution Controls			
Enable Joule Heat Source	Current Under-	Relaxation 0.8		
Enable E-Chem Heat Source	Voltage Correction Under-	Relaxation 1		
<ul> <li>Electrical Parameters</li> </ul>				
Nominal Cell Capacity (ah) 14.6				
Solution Options	C-Rate	1		
Specified C-Rate	System Current (a)	1		
<ul> <li>Specified System Current</li> <li>Specified System Voltage</li> </ul>	System Voltage (v)			
<ul> <li>Specified System Power</li> </ul>	System Power (w)			
Specified Resistance	External Resistance (ohm)			
O Using Profile				
Set in Boundary Conditions	Min. Stop Voltage (v)			
Max. Stop Voltage (v) 4.3				
OK Init Reset Apply Cancel Help				

- b. Under the **Model Options** tab (Figure 24.2: Model Options (p. 965)), configure the following battery operation conditions:
  - i. Under E-Chemistry Models, retain the default selection of NTGK Empirical Model.
  - ii. Ensure that Solving Transport Equation is selected for Solution Method for E-field.
  - iii. Under Electrical Parameters, retain the default value of 14.6 Ah for Nominal Cell Capacity.
  - iv. Retain the default selection of **Specified C-Rate** and the value of 1 for **C-Rate**.
  - v. Retain the default value of 3 V for Min. Stop Voltage.

c. Under the **Model Parameters** tab, retain the default settings for Y and U coefficients. For details, see the Fluent Advanced Add-On Modules documentation on the ANSYS Customer Portal (http://support.an-sys.com/documentation).

## Note

- If in your case, Y and U functions are not in the same function form as in the Kim's paper, you need to modify the cae\_user.c source code file. See the Fluent Advanced Add-On Modules documentation on the ANSYS Customer Portal (http://support.ansys.com/documentation) for details.
- For a given battery, you can perform a set of constant current discharging tests, and then use the battery's parameter estimation tool to obtain the Y and U functions. See the Fluent Advanced Add-On Modules documentation on the ANSYS Customer Portal (http://sup-port.ansys.com/documentation) for details.
- d. Under the **Conductive Zones** tab (Figure 24.3: Conductive Zones (p. 967)), configure the following settings:

Group	Control or List	Value or Selection
Active Components	Zone (s)	e-zone
Tab Components	Zone (s)	tab_nzone
		tab_pzone

For this single cell case, there are no busbar zones. Electro-chemical reactions occur only in the active zone. Battery tabs are usually modeled as passive zones, in which the potential field is also solved.

<b>_</b>						
1	MSMD Battery N	Model				×
	Enable MSMD I	Battery Model				
	Model Options	Model Parameters	Conductive Zones	Electric Conta	cts Advanced Option	n
	- Active Compone	ents	Tab Components		Busbar Components	]
	Zone(s) [1/3]	<b></b>	Zone(s) [2/3]		Zone(s) [0/3]	
	e_zone tab_nzone tab_pzone		e_zone tab_nzone tab_pzone		e_zone tab_nzone tab_pzone	
	OK Init Reset Apply Cancel Help					

## Figure 24.3: Conductive Zones

e. Under the **Electric Contacts** tab (Figure 24.4: Electric Contacts (p. 968)), configure the contact surface and external connector settings as follows:

Group	Control or List	Value or Selection
External Connectors	Negative Tap	tab_n
	Positive Tap	tab_p

The corresponding current or voltage boundary condition will be applied to those boundaries automatically.

Under the **Electric Contacts** tab, you can also define extra contact resistance for each zone.

## Figure 24.4: Electric Contacts

SMD Battery	Model				<b>—</b> × <b>—</b>
Enable MSMD	Battery Model				
Model Options	Model Parameters	Conductive Zones	Electric Contacts	Advanced Option	
- Contact Surface	25				
Zone(s) [0/5]				Contact Resistance	e (ohm-m2)
tab_n					
tab_p wall_active					
wall_n					
wall_p					
	tery connection				
- External Connec					
Negative Tab [1,	/5]		sitive Tab [1/5]		-7 -x
tab_n			b_n		
tab_p wall_active			ıb_p all_active		
wall_n		w	all_n		
wall_p		W	all_p		
Drint Bathany Gu					
Print Battery System Connection Information					
	ОК	Init Reset Ap	ply Cancel Help		

## f. Click the **Print Battery System Connection Information** button.

ANSYS Fluent prints the battery connection information in the console window:

Battery Network Zone	Information:					
Battery Serial 1 Parallel 1						
N-Tab zone:	tab_nzone					
Active zone:	e_zone					
P-Tab zone:	tab_pzone					
Active zone list at Active zone list at	_					

- g. Verify that the connection information is correct.
- h. Click OK to close the MSMD Battery Model dialog box.

In the background, Fluent automatically hooks all the necessary UDFs for the problem.

## 24.4.4.2. Defining New Materials for Cell and Tabs

Define the new e\_material material for the battery's cell, p\_material for the positive tab, and n\_material for the negative tab.

In the battery model, two user defined scalars, uds0 and uds1, are solved for the positive and negative potentials, respectively. To specify the electric conductivity of the active material you need to define the UDS diffusivity.

### Important

For the battery active material, you must define the electric conductivity via the UDS diffusivity on a per-scalar basis. ANSYS Fluent will use these two UDS scalars to solve the differential equations; for details, see the Fluent Advanced Add-On Modules documentation on the ANSYS Customer Portal (http://support.ansys.com/documentation).

For the battery passive conductive zones, such as tabs or busbars, you must define the UDS diffusivity using a user-defined function (UDF) and define the material's electric conductivity using the **Electrical Conductivity** entry field.

1. Create the electric material.

Setting Up Physics  $\rightarrow$  Materials  $\rightarrow$  Create/Edit...

Create/Edit Materials	Material Trans	Order Materials by
e_material	Material Type solid	<ul> <li>Name</li> </ul>
Chemical Formula	Fluent Solid Materials	Chemical Formula
e	e_material (e)	<ul> <li>Fluent Database</li> </ul>
	Mixture	User-Defined Database
	none	Ŧ
Properties		
Density (kg/m3)	constant	
	2092	
Cp (Specific Heat) (j/kg-k)	constant	
	678 🗉	
Thermal Conductivity (w/m-k)	constant 👻 Edt	
	18.2	
UDS Diffusivity (kg/m-s)	defined-per-uds	
Ch	ange/Create Delete Close Help	

- a. In the Create/Edit Materials dialog box, select solid from the Material Type drop-down list.
- b. Enter e\_material for Name and e for Chemical Formula.
- c. Under **Properties**, set **Density** to **2092** [kg/m<sup>3</sup>].
- d. Set CP (Specific Heat) to 678 [J/kg-K].
- e. Set Thermal Conductivity to 18.2 [W/m-K].
- f. Ensure that **define-per-uds** is selected from the **UDS Diffusivity** drop-down list and click **Edit...** next to **UDS Diffusivity**.

#### Note

If the UDS Diffusion Coefficients are defined through the defined-per-uds option, the Fluent solver does not use the value for **Electrical Conductivity**.

g. In the UDS Diffusion Coefficients dialog box, specify the user-defined scalars.

UDS Diffusion Coefficients
User-Defined Scalar Diffusion
uds-0 uds-1
<u> </u>
Coefficient (kg/m-s)
constant 👻 Edit
1190000
OK Cancel Help

- i. Select uds-0 in the User-Defined Scalar Diffusion list.
- ii. Retain **constant** from the **Coefficient** drop-down list.
- iii. Set **Coefficient** to 1.19e6 [1/ohm-m].
- iv. In a similar way, set **uds-1** to **9.83e5** [1/ohm-m] and close the **UDS Diffusion Coefficients** dialog box.

### Note

The units for UDS Diffusivity are 1/ohm-m, and cannot be modified.

v. In the **Question** dialog box, click **No** to retain **aluminum** and add the new material (e\_material) to the materials list.

### Note

Refer to Appendix (p. 1001) for information on how to calculate the battery cell property values.

- h. Ensure that e\_material (e) is selected from the Fluent Solid Materials drop-down list.
- i. Click Change/Create.
- 2. Create a new material for the positive tab by modifying copper from the solid material database.

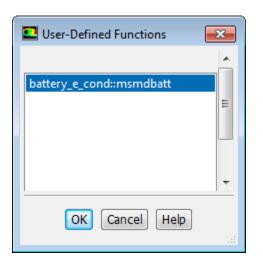
Create/Edit Materials		<b>—</b>
Name	Material Type	Order Materials by
p_material	solid	O Name
Chemical Formula	Fluent Solid Materials	Chemical Formula
pmat	p_material (pmat)	<ul> <li>Fluent Database</li> </ul>
	Mixture	User-Defined Database
	none	T
Properties		
Cp (Specific Heat) (j/kg-k)	constant	
	381	
Thermal Conductivity (w/m-k)	constant	
	387.6	
UDS Diffusivity (kg/m-s)	user-defined	
	battery_e_cond::msmdbatt	
Electrical Conductivity (1/ohm-m)	constant   Edit	
	1e+07	
Ch	inge/Create Delete Close Help	

The UDS Diffusivity for tab materials must be defined through the UDF as described below.

- a. In the Create/Edit Materials dialog box, click Fluent Database....
- b. In the Fluent Database Materials dialog box, make sure that solid is selected for Material Type.
- c. Select copper from Fluent Solids Materials and click Copy and then Close.

The Create/Edit Materials dialog box now displays the copied properties for copper.

- d. Enter p\_material for Name and pmat for Chemical Formula.
- e. Under Properties, select user-defined from the UDS Diffusivity drop-down list.
- f. In the User-Defined Functions dialog box that opens, verify that **battery\_e\_cond::msmdbatt** is selected and click **OK**.



Fluent hooks the battery\_e\_cond::msmdbatt UDF to the busbar material.

g. In the **Question** dialog box, click **Yes** to overwrite **copper**.

The new material (p\_material) appears under Materials.

- h. Set the Electrical Conductivity to 1.0e7 [1/ohm-m].
- i. Click Change/Create.
- 3. Create a new material for the negative tab with the same properties as the material for the positive tab.

### Note

You do not need to create two different materials for the positive and negative tabs if the positive and negative tabs are made of the same material. In this tutorial, the two different tab materials with the same physical properties have been created for demonstration purposes only.

- a. From Fluent Solid Materials drop-down list, select p\_material.
- b. Enter n\_material for Name and nmat for Chemical Formula.
- c. Click Change/Create.
- d. In the **Question** dialog box, click **No** to retain p\_material and add the new material (n\_material) to the materials list.
- e. Close the Create/Edit Materials dialog box.

## 24.4.4.3. Defining Cell Zone Conditions

Assign e\_material to the cell zone, p\_material to the positive tab and n\_material to the negative tab.

1. Assign e\_material to the e\_zone zone.

**Setup**  $\rightarrow$  **Cell** Zone Conditions  $\rightarrow \stackrel{\frown}{=} e_zone \rightarrow Edit...$ 

a. In the **Solid** dialog box, select e\_material from the **Material Name** drop-down list.

- b. Click OK.
- 2. In a similar manner, assign p\_material to tab\_pzone and n\_material to tab\_nzone.

## 24.4.4.4. Defining Boundary Conditions

Define the thermal boundary conditions for all walls for the cell, and positive and negative tabs.

1. Set the convection boundary condition for wall\_active.

**Setup**  $\rightarrow$  **Conditions**  $\rightarrow \stackrel{\bullet}{=}$  wall\_active  $\rightarrow$  Edit...

- a. In the Wall dialog box, under the Thermal tab, under Thermal Conditions, enable Convention.
- b. Set Heat Transfer Coefficient to 5  $[w/m^2K]$ .
- c. Retain the default value of 300 [K] for Free Stream Temperature.
- d. Click **OK** to close the **Wall** dialog box.

You do not need to change the settings under the UDS tab since the boundary conditions for the two UDS scalars have been set automatically when you defined the cell zone conditions.

2. Copy the boundary conditions for wall\_active to wall\_p and wall\_n.

# **Setup** $\rightarrow$ **Copy... Setup** $\rightarrow$ **Copy...**

Copy Conditions	
From Boundary Zone Filter Text	To Boundary Zones Filter Text
default-interior:012	<ul> <li>tab_n</li> </ul>
default-interior:013	tab_p
default-interior:014	wall_n
tab_n	wall_p
tab_p	E
wall_active	
wall_n	
wall_p	•
	Copy Close Help

## 24.4.4.5. Specifying Solution Settings

1. Turn off the flow equation.



- a. In the **Equations** dialog box, deselect Flow from the **Equation** selection list.
- b. Click OK.

2. Remove the convergence criteria to ensure that automatic convergence checking does not occur.

**Solving**  $\rightarrow$  Reports  $\rightarrow$  Residuals...

- a. In the Residual Monitors dialog box, select none from the Convergence Criterion drop-down list.
- b. Click **OK**.
- 3. Create a surface report definition for the voltage at the positive tab.

**Solving**  $\rightarrow$  Reports  $\rightarrow$  Definitions  $\rightarrow$  New  $\rightarrow$  Surface Report  $\rightarrow$  Area-Weighted Average

Surface Report Definition		
Name	Report Type	
voltage_vp	Area-Weighted Average	
Options	Custom Vectors	
	Vectors of	
Per Surface	current-density-jn	
Average Over	Custom Vectors	
1	Field Variable	
	User Defined Scalars	
Report Files [0/0]		
	Potential Phi+	
	Surfaces Filter Text	
	default-interior	
	default-interior:010	
Report Plots [0/0]	default-interior:012	
	default-interior:013 default-interior:014	
	tab_n	
	tab_p	
	wall_active	
Create	wall_n	
Report File	wall_p	
Report Plot		
Frequency 1 🚔		
Print to Console	Highlight Surfaces	
Create Output Parameter New Surface 🔻		
OK Compute Cancel Help		

- a. In the **Surface Report Definition** dialog box, enter **voltage\_vp** for **Name**.
- b. Select User Defined Scalars... and Potential Phi+ from the Field Variable drop-down lists.
- c. From the **Surfaces** selection list, select **tab\_p**.

- d. In the Create group box, enable Report File, Report Plot and Print to Console.
- e. Click **OK** to save the **voltage\_vp** report definition and close the **Surface Report Definition** dialog box.
- f. Rename the report output file.

Solution  $\rightarrow$  Monitors  $\rightarrow$  Report Files  $\rightarrow$  voltage\_vp-rfile  $\stackrel{\frown}{\rightarrow}$  Edit...

Edit Report File	
Name	
voltage_vp-rfile	
Available Report Definitions [0/3]	Selected Report Definitions [0/1]
delta-time flow-time iters-per-timestep	Add>> < <remove< td=""></remove<>
Output File Base Name ntgk-1c.out Browse Full File Name Get Data Every 1 + time-step V Print to Console	New V Edit
OK	Cancel Help

- i. Enter ntgk-1c.out for Output File Base Name.
- ii. Click **OK** to close the **Edit Report File** dialog box.
- g. Modify the attributes of the plot axes.

Solution → Monitors → Report Plots → voltage\_vp-rplot  $\stackrel{\frown}{\hookrightarrow}$  Edit...

- i. In the **Edit Report Plot** dialog box, under the **Plot Window** group box, click the **Axes...** button to open the **Axes** dialog box.
- ii. Select the X axis and set Precision to 0.
- iii. Click Apply.
- iv. Select the Y axis and set Precision to 2.
- v. Set **Precision** to 2.

vi. Click **Apply** and close the **Axes** dialog box.

Note

You must click **Apply** to save the modified settings for each axis.

vii. Make sure that **time-step** is selected from the **Get Data Every** drop-down list.

viii. Click OK to close the Edit Report Plot dialog box.

4. Create a volume report definition for the maximum temperature in the domain.

Solving  $\rightarrow$  Reports  $\rightarrow$  Definitions  $\rightarrow$  New  $\rightarrow$  Volume Report  $\rightarrow$  Max...

Volume Report Definition	<b>—</b>	
Name	Report Type	
max_temp	Max	
Options	Field Variable	
	Temperature 🔻	
Per Zone	Static Temperature 🗸 🗸	
Average Over		
1	Cell Zones Filter Text	
	e_zone	
Report Files [0/1]	-	
voltage_vp-rfile	tab_pzone	
Report Plots [0/0]		
Create		
📝 Report File		
📝 Report Plot		
Frequency 1		
Print to Console		
Create Output Parameter		
OK Compute Cancel Help		

a. In the **Volume Report Definition** dialog box, enter max\_temp for Name.

- b. Select Temperature... and Static Temperature from the Field Variable drop-down lists.
- c. From the **Cell Zones** selection list, select all zones.
- d. In the Create group box, enable Report File, Report Plot and Print to Console.
- e. Click **OK** to save the volume report definition settings and close the **Volume Report Definition** dialog box.
- f. Rename the report output file.

**T** Solution  $\rightarrow$  Monitors  $\rightarrow$  Report Files  $\rightarrow$  max\_temp-rfile  $\stackrel{\square}{\hookrightarrow}$  Edit...

- i. Enter max-temp-1c.out for Output File Base Name.
- ii. Click **OK** to close the **Edit Report File** dialog box.
- g. Modify the axis attributes by setting the **Precision** to 0 for the **X** axis and to 2 for the **Y** axes (in a manner similar to the surface plot definition).
- h. Click OK.
- 5. Save the case file.

**File**  $\rightarrow$  Write  $\rightarrow$  Case...

## 24.4.4.6. Obtaining Solution

1. Initialize the field variables using the **Standard Initialization** method.

Solving → Initialization

- a. Retain the selection of the Standard method (Initialization group).
- b. Click Initialize.

You do not need to modify **Initial Values** in the **Solution Initialization** task page, because these values are not used for initialization. The ANSYS Fluent solver automatically computes the initial condition for UDS0 and UDS1.

2. Run the simulation.

## Solving → Run Calculation

- a. Set Time Step Size to 30 seconds and No. of Time Steps to 100.
- b. Click Calculate.

The residual plot, the report for voltage at the positive tap and the history of the maximum temperature in the domain are shown in Figure 24.5: Residual History of the Simulation (p. 979), Figure 24.6: Report Plot of Discharge Curve at 1 C (p. 979), and Figure 24.7: History of Maximum Temperature in the Domain (p. 980), respectively.

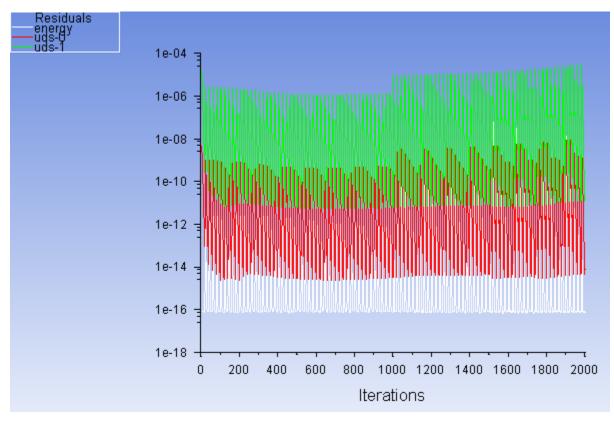
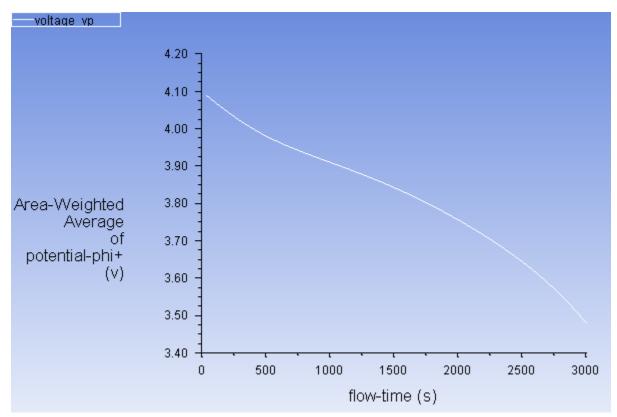


Figure 24.5: Residual History of the Simulation





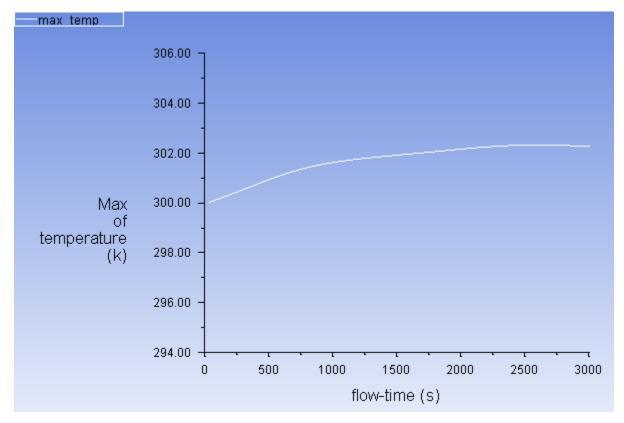


Figure 24.7: History of Maximum Temperature in the Domain

3. Save the case and data files (unit\_battery.cas.gz and unit\_battery.cas.gz).

File  $\rightarrow$  Write  $\rightarrow$  Case & Data...

## 24.4.5. Postprocessing

In this section, postprocessing capabilities for the MSMD battery model solution are demonstrated.

1. Display the contour plot of the phase potential for the positive electrode.



Contours	
Options Filled	Contours of User Defined Scalars
<ul> <li>Node Values</li> <li>Global Range</li> </ul>	Potential Phi+
Auto Range	Min Max 0 0
Draw Profiles	Surfaces Filter Text
	default-interior:014
Coloring Banded Smooth	tab_n tab_p wall_active
Levels Setup	wall_active E wall_n Vall_p T
20 🌩 1 🌩	New Surface
	Display Compute Close Help

- a. In the **Contours** dialog box, in the **Options** group box, enable **Filled**.
- b. From the Contours of drop-down list, select User Defined Scalars... and Potential Phi+.
- c. Click the **Toggle Tree View** button next to the **Surfaces** filter and from the drop-down list, select **Surface Type** (under **Group by**).
- d. From the **Surfaces** selection list, under **Wall**, select **tab\_p**, **wall\_active**, and **wall\_p**.
- e. Click **Display** and close the **Contours** dialog box.

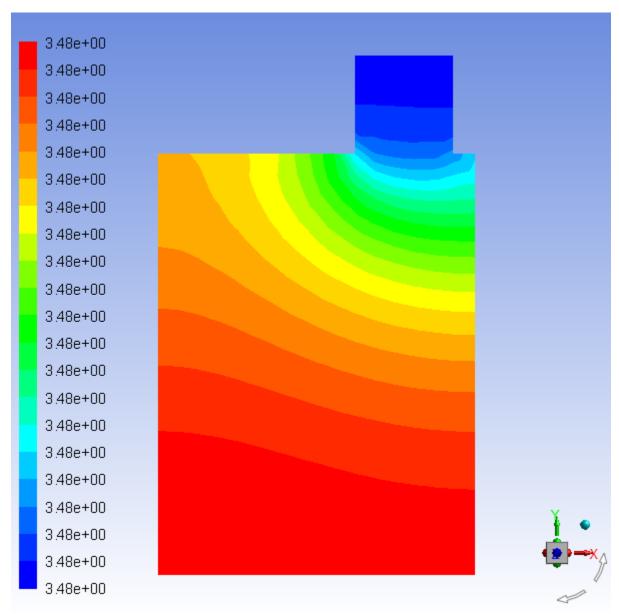


Figure 24.8: Contour Plot of Phase Potential for the Positive Electrode

2. In a similar manner, display the contour plot of the phase potential for the negative electrode.

Postprocessing  $\rightarrow$  Graphics  $\rightarrow$  Contours  $\rightarrow$  Edit...

Contours	
Options Filled Node Values Global Range Auto Range	Contours of User Defined Scalars
	Potential Phi-     ▼       Min (v)     Max (v)
<ul> <li>Clip to Range</li> <li>Draw Profiles</li> <li>Draw Mesh</li> </ul>	-0.003310583 0 Surfaces Filter Text
Coloring Banded Smooth Levels Setup 20 1	default-interior:014  Wall tab_n tab_p wall_active wall_n wall_p  New Surface
	Display Compute Close Help

- a. In the **Contours** dialog box, in the **Options** group box, enable **Filled**.
- b. From the Contours of drop-down list, select User Defined Scalars... and Potential Phi-.
- c. From the **Surfaces** selection list, select tab\_n, wall\_active, and wall\_n.
- d. Click **Display** and close the **Contours** dialog box.

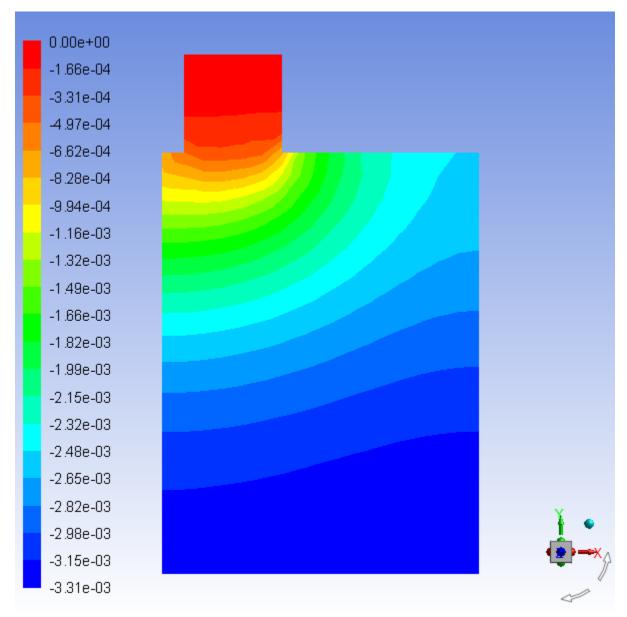


Figure 24.9: Contour Plot of Phase Potential for the Positive Electrode

3. Display the contour plot of the temperature.

Postprocessing  $\rightarrow$  Graphics  $\rightarrow$  Contours  $\rightarrow$  Edit...

Contours		<b>.</b>
Options	Contours of	
Filled		
Node Values	Static Temperature	<b>•</b>
<ul> <li>Global Range</li> <li>Auto Range</li> </ul>	Min (k)	Max (k)
Clip to Range	301.504	301.9125
Draw Profiles	Surfaces Filter Text	
Coloring Banded Smooth Levels Setup 20 1	✓ Wall tab_n tab_p wall_active wall_n wall_p	

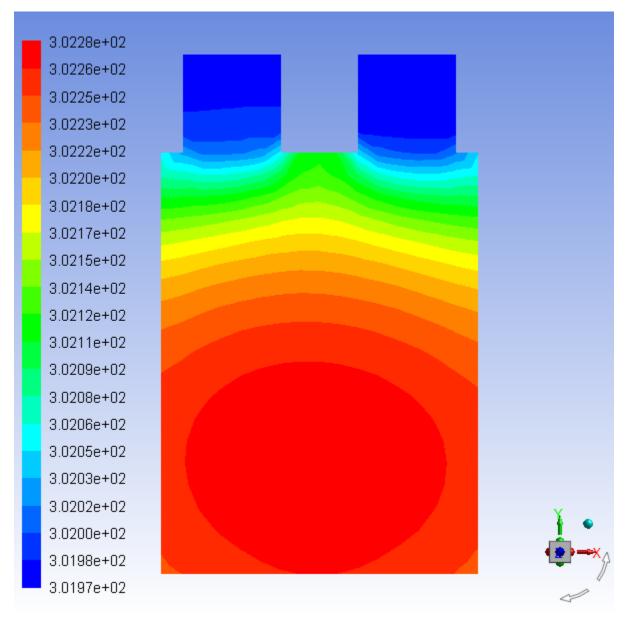
- a. In the **Contours** dialog box, in the **Options** group box, enable **Filled**.
- b. From the **Contours of** drop-down list, select **Temperature...** and **Static Temperature**.
- c. Select **Wall** in the **Surfaces** selection list.

The surfaces listed under **Wall** are automatically selected in the **Surfaces** list.

d. Click **Display** and close the **Contours** dialog box.

### Note

Use the **Axes** dialog box to set the precision for the colormap labels.



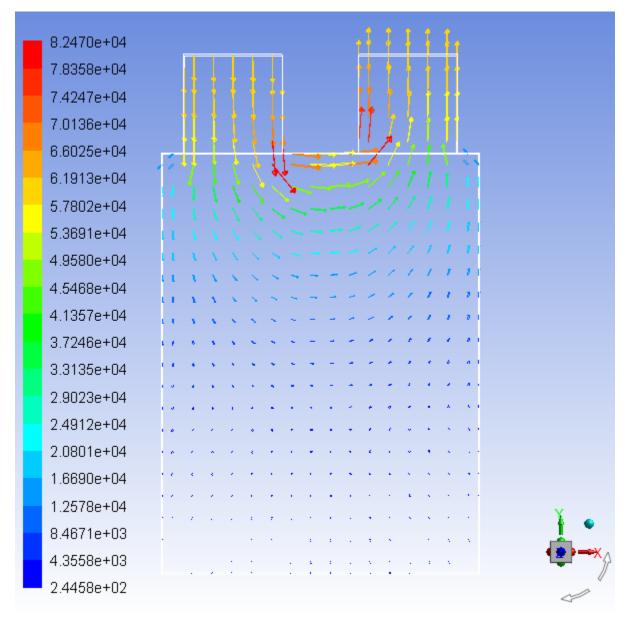
## Figure 24.10: Contour Plot of Temperature

4. Display the vector plot of current density.

Postprocessing  $\rightarrow$  Graphics  $\rightarrow$  Vectors  $\rightarrow$  Edit...

Vectors		<b>X</b>
Options	Vectors of	
Global Range	current-density-j	▼]
Auto Range	Color by	
Clip to Range	User Defined Memory	/ <b>*</b> ]
Auto Scale Craw Mesh	Magnitude of Curren	t Density 👻
	Min (a/m2)	Max (a/m2)
Style	244.5763	82469.23
arrow 👻 Scale Skip	Surfaces Filter Text	<b>F</b> . <b>F</b> . <b>F</b> .
1 0 🚔	default-interio	r:014
Vector Options	▲ Wall	
Custom Vectors	tab_n tab_p	
	wall_active	Ξ.
	wall_n	-
	New Surface 🔻	
	Display Compute	Close Help

- a. In the Vectors dialog box, select current-density-j from the Vectors of drop-down list.
- b. Select User Defined Memory... and Magnitude of Current Density from the Color by drop-down list.
- c. Click the **Toggle Tree View** button next to the **Surfaces** filter and from the drop-down list, select **Surface Type** (under **Group by**).
- d. From the **Surfaces** selection list, select **Wall**.
- e. In the **Options** group, enable **Draw Mesh** and in the **Mesh Display** dialog box, set the mesh display options as desired.
- f. Click **Display** and close the **Vectors** dialog box.



## Figure 24.11: Vector Plot of Current Density

- 5. Save the case file as ntgk.cas.gz. You will use this saved case later to treat electric short-circuits.
- 6. Repeat the simulation for the following charge rates and time steps:

C-Rate	Number of Time Steps
0.5 C	230
5 C	23

Make the following changes in the model's settings:

- a. In the **MSMD Battery Model** dialog box, under the **Model Options** tab, specify the value listed in the above table for the **C-Rate**.
- b. Modify the output filename for the **voltage\_vp-rfile** report file by entering **ntgk-***C***-***Rate***<b>.out** for **Output File Base Name** in the corresponding **Edit Report File** dialog box, where *C*-*Rate* **is the value**

of the battery discharge rate. (For example, for C-Rate = 0.5 C, you will enter **ntgk-0.5c.out** for the filename).

- c. Similarly, modify the output filename for **max\_temp-rfile** by entering **max-temp-***C***-***Rate.***out** for **Output File Base Name** in the corresponding **Edit Report File** dialog box.
- d. Initialize and run the solution for the number of the times steps specified in the above table.

#### Note

The Fluent solver will stop either after completing the specified number of time steps or when the **Min. Stop Voltage** condition is reached.

7. Display the discharge curves for the positive tab for the different discharge rates.

## Postprocessing $\rightarrow$ Plots $\rightarrow$ File...

- a. In the File XY Plot dialog box, click Add....
- b. In the Select File dialog box, change File of Types to All Files (\*), click ntgk-0.5c.out and click OK.
- c. Under Legend Entries, in the lowest text-entry box, enter 0.5c and click Change Legend Entry.
- d. Do the same for ntgk-1c.out and ntgk-5c.out and change their legend entries accordingly.
- e. In the **Legend Title** text-entry box, enter **Discharge Rate**.
- f. Click **Plot** and close the **File XY Plot** dialog box.

#### Note

Use the Axes dialog box to set the precision for the plot axes.

The Figure 24.12: NTGK Model: Discharge Curves (p. 990) shows the discharge curves for different discharge rates in the function of time.

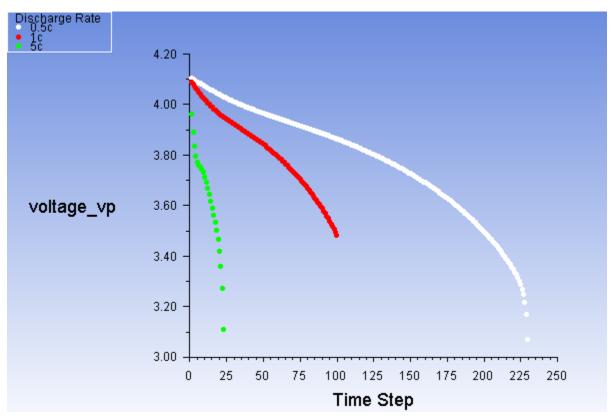
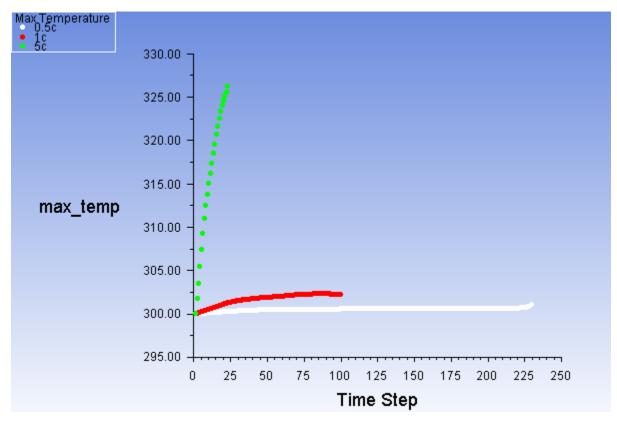


Figure 24.12: NTGK Model: Discharge Curves

8. In a manner similar to the previous step, load the files max-temp-0.5c.out, max-temp-1c.out, and max-temp-5c.out and display the maximum temperature curves in the domain.

Figure 24.13: NTGK Model: Maximum Temperature in the Domain (p. 991) shows the maximum temperature curves in the simulation for different discharge rates.



### Figure 24.13: NTGK Model: Maximum Temperature in the Domain

# 24.4.6. Simulating the Battery Pulse Discharge Using the ECM Model

- 1. In the MSMD Battery Model dialog box, under E-Chemistry Models, select Equivalent Circuit Model.
- 2. Under Electrical Parameters, retain the default value of 14.6 Ah for Nominal Cell Capacity.
- 3. Retain the default selection of **Specified C-Rate** and enter 1 for **C-Rate**.
- 4. Under the Model Parameters tab, retain the battery specific parameters.

For a given battery, these model parameters can be obtained using the battery's HPPC testing data. See Using the Dual-Potential MSMD Battery Model Text User Interface in the ANSYS Fluent Advanced Add-On Modules for details.

- 5. Click **OK** to apply the ECM battery model settings and close the **MSMD Battery Model** dialog box
- 6. Click **OK** in the **Warning** dialog box informing you that the re-initialization of the battery model is required.
- 7. Disable writing the maximum temperature in the domain over time to a file.

**Solutions**  $\rightarrow$  Report Definitions  $\rightarrow$  max\_temp  $\stackrel{\bigoplus}{\rightarrow}$  Edit...

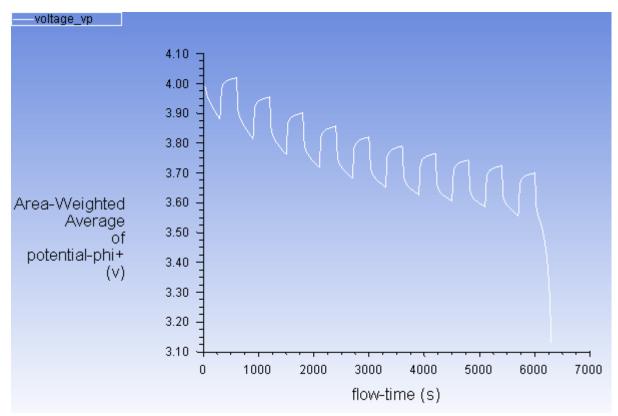
- a. In the Volume Report Definition dialog box, under Report Files, deselect max\_temp-rfile.
- b. Click **OK** to close the **Volume Report Definition** dialog box.
- 8. In a similar manner, disable writing the time-dependent voltage at the positive tab to a file.

- 9. In the Solution Initialization task page, click Initialize to re-initialize the field variables.
- 10. Simulate the battery pulse discharge by changing the battery operating conditions each time after running the calculation for five minutes.
  - a. In the Run Calculation task page, set Number of Time Steps to 10 and click Calculate.
  - b. Once the calculation is complete, set **C-Rate** to 0 and run the calculation for 10 more time steps.
  - c. Continue the simulation by alternating the value of **C-Rate** between 1 C and 0 C until, until the battery is fully discharged.

### Note

Instead of doing this manually, you can use the **Using Profile** option in the **MSMD Battery Model** dialog box and load a profile file with specified C-rate fluctuations to drive the whole process. For more information about the usage of a profile file, refer to Specifying Battery Model Options in the ANSYS Fluent Advanced Add-On Modules..

The battery pulse discharge is summarized in Figure 24.14: Battery Pulse Discharge (p. 992).



### Figure 24.14: Battery Pulse Discharge

# 24.4.7. Using the Reduced Order Method (ROM)

You will use the ntgk.cas.gz case file that you saved earlier to illustrate how to use the ROM for time-efficient calculations. This section assumes that you are already familiar with the ANSYS Fluent

battery model; only the steps related specifically to using the ROM for problem solution are discussed here.

- 1. Read the NTGK model case file ntgk.cas.gz.
- 2. Initialize the problem.
- 3. In the Run Calculation task page, enter 3 for Number of Time Steps and click Calculate.

Click **No** in the **Question** dialog box when asked if you would like to append the new data to the existing file, and then click **OK** in the **Warning** dialog box to overwrite the existing file.

4. Once the calculation is complete, enable the ROM.

```
Example : Setup \rightarrow Models \rightarrow MSMD Battery Model \stackrel{\frown}{\subseteq} Edit...
```

- a. In the MSMD Battery Model dialog box, select Reduced Order Method.
- b. Set Number of Sub-Steps/Time Step to 10.
- 5. Continue running your simulation from the direct method solution.

The solution of the simulation using the ROM is significantly faster than when using the direct method without any changes in results.

## 24.4.8. External and Internal Short-Circuit Treatment

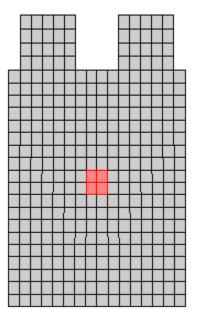
You will again use the ntgk.cas.gz case file that you saved earlier to illustrate how to treat external and internal short-circuits in a battery simulation. It is assumed that the battery is experiencing external and internal short-circuit simultaneously. This extreme case will be used to demonstrate the problem setup and postprocessing in a short simulation. This section assumes that you are already familiar with the ANSYS Fluent battery model, only the steps related to short simulation are emphasized here.

## 24.4.8.1. Setting up and Solving a Short-Circuit Problem

- 1. Read the NTGK model case file ntgk.cas.gz.
- 2. Set up the external electric short-circuit.
  - a. In the **MSMD Battery Model** dialog box, under the **Model Options** tab, in the **Solution Options** group box, enable **Specified Resistance**.
  - b. For External Resistance, enter 0.5 Ohm and click OK.
- 3. Set up the internal electric short-circuit in the center of the battery cell.
  - a. Mark the short-circuit zone shown in Figure 24.15: Internal Short Circuit Region Marked for Patching (p. 994) using the region adaption feature.

Setting Up Domain  $\rightarrow$  Adapt  $\rightarrow$  Mark/Adapt Cells  $\rightarrow$  Region...

#### Figure 24.15: Internal Short Circuit Region Marked for Patching



i. In the **Region Adaption** dialog box, enter the following values for **Input Coordinates**.

X Min	X Max	Y Min	Ү Мах	Z Min	Z Max
-0.01	0.01	-0.01	0.02	-1	1

- ii. Under **Options**, retain the default selection of **Inside**.
- iii. Under **Shapes**, retain the default selection of **Hex**.
- iv. Click Mark.

Fluent reports in the console that 12 cells were marked for refinement.

#### Extra

If you want to display the marked cells, perform the following steps:

- A. In the Region Adaption dialog box, click Manage....
- B. In the Manage Adaption Registers dialog box, click Options.
- C. In the Adaption Display Option dialog box, under Options, enable Draw Mesh.
- D. In the **Mesh Display** dialog box, ensure that **Edges** and **Faces** are enabled under **Options**, and **All** is enabled under **Edge Type**, and click **Close**.
- E. Ensure that Wireframe is enabled under Refine and click OK
- F. Click Display.

- 4. Initialize the field variables using the standard initialization method.
- 5. Patch the internal short circuit zone with the short resistance value.

	Value 5.0e-7	Zones to Patch Filter Text
Absolute	Use Field Function	e_zone tab_nzone
Z Current Density for phi+ X Current Density for phi- Y Current Density for phi- Z Current Density for phi- Short Circuit Resistance Volumetric Short Current Source Volumetric ECHEM Current Source Short-Circuit Heat Source Saved ROM Ref mbia	Field Function	Registers to Patch [1/1]
	Patch Clos	e Help

### **Solving** $\rightarrow$ Initialization $\rightarrow$ Patch...

- a. In the Solution Initialization task page, click Patch.
- b. In the Patch dialog box, select Short Circuit Resistance under Variable.
- c. Select hexehedron-r0 under Registers to Patch.
- d. For Value, enter 5.0e-7.
- e. Click Patch and close the Patch dialog box.
- 6. Save the case file as ntgk\_short\_circuit.cas.gz.
- 7. Run the simulation for 5 seconds.

## Solving → Run Calculation

- a. Set Time Step Size to 1 second and No. of Time Steps to 5.
- b. Click Calculate.
- 8. Save the case and data files (ntgk\_short\_circuit.cas.gz and ntgk\_short\_circuit.dat.gz).

### 24.4.8.2. Postprocessing

1. Compute the battery tab voltage  $U_{tab}$ .

Postprocessing  $\rightarrow$  Reports  $\rightarrow$  Surface Integrals...

Surface Integrals	<b>•••</b>			
Report Type	Field Variable			
Area-Weighted Average 🔹	User Defined Scalars			
Custom Vectors	Potential Phi+			
Vectors of current-density-jn	Surfaces t 🗙 📻 🗮 🐺			
Custom Vectors	tab_n			
Save Output Parameter	tab_p			
	<ul> <li>Highlight Surfaces</li> <li>Area-Weighted Average (v)</li> <li>4.077621</li> </ul>			
Compute Write Close Help				

- a. In the **Surface Integrals** dialog box, from the **Report Type** drop-down list, select **Area-Weighted Average**.
- b. From the Field Variable drop-down lists, select User Defined Scalar... and Potential Phi+.
- c. In the **Surfaces** filter, type t to display surface names that begin with "t" and select **tab\_p** from the selection list.
- d. Click Compute.

The battery tab voltage of approximately 4.078 V is printed in the **Area-Weighted Average** field and in the Fluent console.

2. Compute the battery tab current  $I_{tab}$ .

Postprocessing  $\rightarrow$  Reports  $\rightarrow$  Volume Integrals...

Volume Integrals					
Report Type	Field Variable	Cell Zones Filter Text			
Mass-Average	User Defined Memory				
Mass Integral	Volumetric Current Source 👻	e_zone			
O Mass		tab_nzone			
O Sum		tab_pzone			
<ul> <li>Minimum</li> <li>Maximum</li> </ul>					
<ul> <li>Volume</li> </ul>	Total Volume Integral (a/m3)(m3)				
<ul> <li>Volume-Average</li> </ul>	8.155242				
Volume Integral	Save Output Parameter				
Compute Write Close Help					

- a. In the **Report Type** group box, select **Volume Integrals**.
- b. From the Field Variable drop-down lists, select User Defined Memory... and Volumetric Current Source.
- c. From the **Cell Zones** selection list, select e\_zone.
- d. Click Compute.

Fluent reports in the **Total Volume Integral** field and in the console that the total volume integral for the volumetric current source is approximately 8.155 A.

The computed values of the battery tab current and voltage satisfy the tab boundary condition  $R_{load} = U_{tab} / I_{tab}$ .

3. Display the vector plot of current at the positive and negative current collectors.

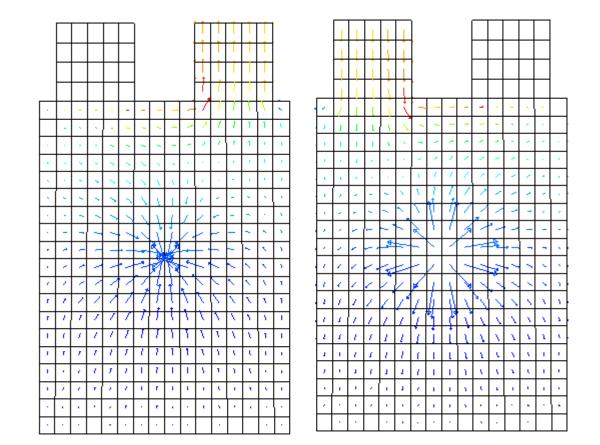
Postprocessing  $\rightarrow$  Graphics  $\rightarrow$  Vectors  $\rightarrow$  Edit...

- a. In the Vectors dialog box, select current-density-jp from the Vectors of drop-down list.
- b. Select User Defined Memory... and Magnitude of Current Density from the Color by drop-down lists.
- c. From the Surfaces selection list, select Wall.

The surfaces of the "wall" type are automatically selected in the Surfaces list.

- d. Click Display.
- e. In a similar manner, display the current for the negative current collector by selecting **current-densityjn** from the **Vectors of** drop-down list.

The Figure 24.16: The Vector Plots of Current at the Positive and Negative Current Collectors (p. 998) demonstrates the vector plots of electric current flow in the positive and negative current collectors in the battery cell side by side. The plots clearly show that besides providing tab current, short current flows form positive electrode to the negative electrode through the short area.

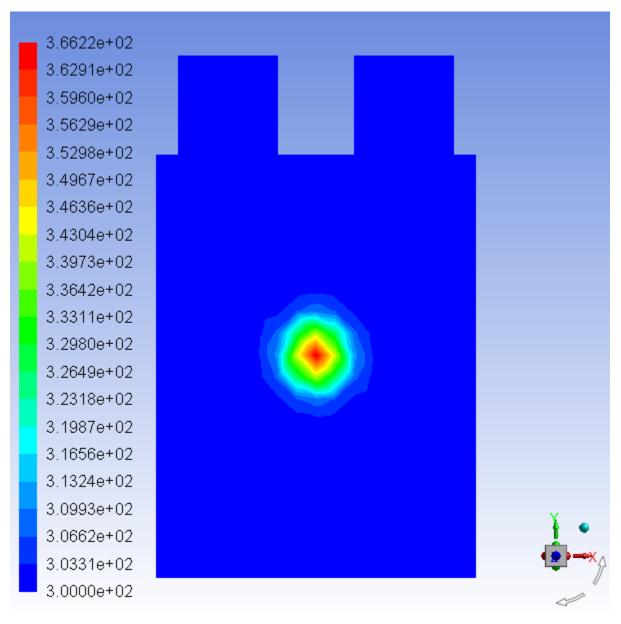


### Figure 24.16: The Vector Plots of Current at the Positive and Negative Current Collectors

- f. Close the **Vectors** dialog box.
- 4. Display the contour plot of the temperature as you did previously.

Contours		-x	
Options Filled	Contours of Temperature		ſ
<ul> <li>Node Values</li> <li>Global Range</li> </ul>	Static Temperature Min (k)	• Max (k)	
Auto Range Clip to Range	300.0004	366.2215	
Draw Profiles Draw Mesh	Surfaces Filter Text		]
Coloring Banded Smooth Levels Setup 20 1	default-interio ✓ Wall tab_n tab_p wall_active wall_n wall_p New Surface ▼ Display Compute	E Close Help	

- a. In the **Contours** dialog box, under **Options**, enable **Filled**.
- b. From the **Contours of** drop-down list, select **Temperature...** and **Static Temperature**.
- c. From the **Surface Types** selection list, select wall.
- d. Click **Display** and close the **Contours** dialog box.



### Figure 24.17: Contour Plot of Temperature

Figure 24.17: Contour Plot of Temperature (p. 1000) shows a temperature hotspot in the internal shorted area of the battery cell.

- 5. Check for different electric current flow rates in the manner described in step 2.
  - a. Generate volume integral reports for the field variables listed in the table below.

Field Variable	Notation	Reported Value	
Volumetric Short Current Source	I <sub>short</sub>	15.844 A	
Volumetric ECHEM Current Source	I <sub>echem</sub>	23.999 A	

b. Verify that the total produced electric current equals to the sum of tab and short current, that is  $I_{echem} = I_{tab} + I_{short}$ .

- 6. Check for different types of heat generation rates.
  - a. As you did for the current source reports, generate reports for the field variables listed in the table below.

Field Variable	Notation	Reported Value
Volumetric Ohmic Source	$Q_{joule}$	0.0339 W
Electrochemistry Source	Q <sub>echem</sub>	1.092 W
Short-Circuit Heat Source	Q <sub>short</sub>	64.607 W
Total Heat Generation Source	$Q_{total}$	65.733 W

b. Verify that the total heat generation rate is the sum of different contributions, that is  $Q_{total} = Q_{ioule} + Q_{echem} + Q_{short}$ .

Note that, as battery's temperature increases, thermal runaway may occur. If thermal runaway starts, some undesirable exothermic decomposition reactions will occur. For thermal runaway simulations, the default electrochemistry model cannot be used. Short treatment can only capture the thermal ramp-up process before the onset of thermal runaway.

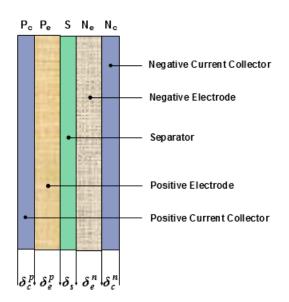
## 24.5. Summary

In this tutorial, you studied how to solve a battery cell problem using the NTGK submodel with the default settings. You then used the ROM to speed up the computation time of the battery model simulation. In addition, you learned how to use the MSMD model capability to treat external and internal short-circuits.

For more information about using the Dual-Potential MSMD Battery model, see the Fluent Advanced Add-On Modules documentation on the ANSYS Customer Portal (http://support.ansys.com/documentation).

# 24.6. Appendix

The battery cell cross-section is shown in the figure below.



You can estimate the material properties for your battery cell using the following correlations:

• For density  $\rho$ , heat capacity  $C_p$ , and thermal conductivity K:

$$x_{eff} = \frac{0.5x_c^p \delta_c^p + x_e^p \delta_e^p + x_s \delta_s + x_e^n \delta_e^n + 0.5x_c^n \delta_c^n}{\delta_{total}}$$
$$\delta_{total} = 0.5\delta_c^p + \delta_e^p + \delta_s + \delta_e^n + 0.5\delta_c^n$$

where  $x_{eff}$  is the effective property value of a material property (such as density, heat capacity, or thermal conductivity), is the thickness. The subscripts c, e, and s refer to current collector, electrode, and separator, respectively. The superscripts p and n refer to positive and negative, respectively.

• For electric conductivity *σ*:

$$\sigma_p = \frac{0.5\sigma_c^p \delta_c^p + \sigma_e^p \delta_e^p}{\delta_{total}}, \quad \sigma_n = \frac{0.5\sigma_c^n \delta_c^n + \sigma_e^n \delta_e^n}{\delta_{total}}$$

The material properties are taken from Kim's papers [2] and [1]. The computed material properties for the battery cell presented in the tutorial are shown in the table below.

Zone	P <sub>c</sub>	P <sub>e</sub>	S	N <sub>e</sub>	N <sub>c</sub>	Total
$\delta$ [um]	20	150	12	145	10	322
ho [kg/m <sup>3</sup> ]	2700	1500	1200	2500	8960	2092
<i>C<sub>p</sub></i> [J/kg-K]	900	700	700	700	385	678
<i>K</i> [W/m-K]	238	5	1	5	398	18.2
σ [s/m]	3.83e7	13.9		100	6.33e7	σ <sub>p</sub> = 1.19e6
						$\sigma_n$ = 9.83e5

# 24.7. References

- 1. U.S. Kim et al, "Effect of electrode configuration on the thermal behavior of a lithium-polymer battery", Journal of Power Sources, Volume 180 (2), pages 909-916, 2008.
- 2. U. S. Kim, et al., "Modeling the Dependence of the Discharge Behavior of a Lithium-Ion Battery on the Environmental Temperature", J. of Electrochemical Soc., Volume 158 (5), pages A611-A618, 2011.

# Chapter 25: Simulating a 1P3S Battery Pack Using the MSMD Battery Model

This tutorial is divided into the following sections: 25.1. Introduction 25.2. Prerequisites 25.3. Problem Description 25.4. Setup and Solution 25.5. Summary

# 25.1. Introduction

This tutorial is used to show how to set up a battery pack (battery system connected in parallel/series pattern) simulation in ANSYS Fluent. All the three submodels are available for a pack simulation.

This tutorial illustrates how to do the following:

- Load the Battery module add-on
- Set up a battery pack simulation using the NTGK battery submodel in ANSYS Fluent
- Define active, tab, and busbar conductive zones
- · Define electric contacts for the contact surface and external connectors
- Define electric conductivity for the active material using the user-defined scalars
- Define electric conductivity for the passive material using the user-defined function
- Obtain the battery pack simulation results and perform postprocessing activities

Most problem setup procedures are similar to the single cell simulation. The differences in the problem setup will be emphasized in this tutorial.

# 25.2. Prerequisites

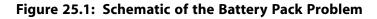
This tutorial is written with the assumption that you have completed one or more of the introductory tutorials found in this manual:

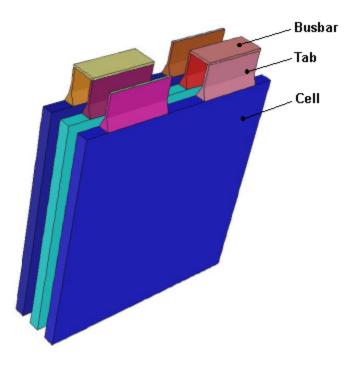
- Introduction to Using ANSYS Fluent in ANSYS Workbench: Fluid Flow and Heat Transfer in a Mixing Elbow (p. 1)
- Parametric Analysis in ANSYS Workbench Using ANSYS Fluent (p. 73)
- Introduction to Using ANSYS Fluent: Fluid Flow and Heat Transfer in a Mixing Elbow (p. 121)

and that you are familiar with the ANSYS Fluent tree and ribbon structure. Some steps in the setup and solution procedure will not be shown explicitly.

# **25.3. Problem Description**

This problem considers a small 1P3S battery pack, that is, the three battery cells connected in series. A schematic of the problem is shown in Figure 25.1: Schematic of the Battery Pack Problem (p. 1006).





The discharging process of the battery pack is occurring under constant power of 200 W. The nominal cell capacity is 14.6 Ah.

You will create a material for the battery cells (an active material) and define the electric conductivity for the active material using the user-defined scalars (UDS). You will create a material for busbars and tabs (a passive material) and define the electric conductivity for the passive material using the provided user-defined function (UDF). You will use the same material for busbars and tabs.

In this tutorial, you will use the NTGK battery submodel to simulate the discharging process under constant power conditions.

# 25.4. Setup and Solution

The following sections describe the setup and solution steps for this tutorial:

- 25.4.1. Preparation
- 25.4.2. Reading and Scaling the Mesh
- 25.4.3. Loading the MSMD battery Add-on
- 25.4.4. Battery Model Setup
- 25.4.5. Postprocessing

# 25.4.1. Preparation

1. Set up a working folder on the computer you will be using.

2. Go to the ANSYS Customer Portal, https://support.ansys.com/training.

#### Note

If you do not have a login, you can request one by clicking **Customer Registration** on the log in page.

- 3. Enter the name of this tutorial into the search bar.
- 4. Narrow the results by using the filter on the left side of the page.
  - a. Click ANSYS Fluent under Product.
  - b. Click **18.0** under **Version**.
- 5. Select this tutorial from the list.
- 6. Click the **battery\_pack\_R180.zip** link to download the input files.
- 7. Unzip battery\_pack\_R180.zip.

The input file 1P3S\_battery\_pack.msh, 1P3S\_battery\_pack.cas, and 1P3S\_battery\_pack.dat can be found in the battery\_pack folder created after unzipping the file.

- 8. Use Fluent Launcher to start the **3D** version of ANSYS Fluent.
- 9. Enable **Double Precision**.
- 10. Ensure that the **Display Mesh After Reading** and **Workbench Color Scheme** options are enabled.
- 11. Select either **Serial** or **Parallel** under **Processing Options**. In this tutorial, **Serial** is selected for demonstration only.

## 25.4.2. Reading and Scaling the Mesh

1. Read the mesh file 1P3S\_battery\_pack.msh.

File  $\rightarrow$  Read  $\rightarrow$  Mesh...

When prompted, browse to the location of the 1P3S\_battery\_pack.msh and select the file.

Once you read in the mesh, it is displayed in the embedded graphics windows.

2. Check the mesh.



3. Scale the mesh.

Setting Up Domain  $\rightarrow$  Mesh  $\rightarrow$  Scale

a. In the Scale Mesh dialog box, select Specify Scaling Factors in the Scaling group.

- b. Enter 0.1 for X, Y and Z in the Scaling Factors group.
- c. Click Scale and close the Scale Mesh dialog box.

Scale M	esh			
-Domain E	xtents			Scaling
Xmin (m)	0	Xmax (m)	0.1	Convert Units
Ymin (m)	-0.016	Ymax (m)	0.1	Specify Scaling Factors
Zmin (m)	-0.0025	Zmax (m)	0.0225	Mesh Was Created In
View Length Unit In m			Scaling Factors X 0.1 Y 0.1 Z 0.1 Scale Unscale	
Close Help				

4. Check the mesh.



# 25.4.3. Loading the MSMD battery Add-on

1. To load the MSMD battery add-on into ANSYS Fluent, type the following in the console window:

define/models/addon-module

A list of ANSYS Fluent add-on modules is displayed.

```
Fluent Addon Modules:
0. none
1. MHD Model
2. Fiber Model
3. Fuel Cell and Electrolysis Model
4. SOFC Model with Unresolved Electrolyte
5. Population Balance Model
6. Adjoint Solver
7. Single-Potential Battery Model
8. Dual-Potential MSMD Battery Model
9. PEM Fuel Cell Model
10. Macroscopic Particle Model
Enter Module Number: [0] 8
```

2. Enter 8 to load the MSMD battery itself.

During the loading process, a scheme library containing the graphical and text user interface, and a library of user-defined functions (UDFs) containing a set of UDFs for the battery module are loaded into ANSYS Fluent. Fluent reports the progress in the console.

Once the MSMD battery add-on is loaded, **MSMD Battery Model** appears in the **Model** task page and under the **Models** tree branch. The UDF library also becomes visible as a new entry in the UDF Library Manager dialog box.

## 25.4.4. Battery Model Setup

The following sections describe the setup steps for this tutorial: 25.4.4.1. Specifying Solver and Models 25.4.4.2. Defining New Materials 25.4.4.3. Defining Cell Zone Conditions 25.4.4.4. Defining Boundary Conditions 25.4.4.5. Specifying Solution Settings 25.4.4.6. Obtaining Solution

## 25.4.4.1. Specifying Solver and Models

1. Enable a time-dependent calculation by selecting **Transient** in the **Solver** group of the **Setting Up Physics** tab.

**Setting Up Physics**  $\rightarrow$  Solver  $\rightarrow$  Transient

2. To solve for the temperature field, enable the **Energy** equation (in the **Models** group).



3. Enable the battery model.

**Setting Up Physics**  $\rightarrow$  Models  $\rightarrow$  More  $\rightarrow$  Dual Potential MSMD Battery Model

a. In the MSMD Battery Model dialog box, select Enable MSMD Battery Model.

The dialog box expands to display the battery model's settings.

### Figure 25.2: Model Options

SMD Battery Model							
Enable MSMD Battery Model							
Model Options Model Parameters Conductive Zones Electric Contacts Advanced Option							
E-Chemistry Models <ul> <li>NTGK Emperical Model</li> <li>Equivalent Circuit Model</li> <li>Newman P2D Model</li> <li>User-defined E-Model</li> </ul>	Solution Method for E-Field Solving Transport Equation Reduced Order Method						
Energy Source Options	Solution Controls						
Enable Joule Heat Source	Current Under-Relaxation 0.8						
Enable E-Chem Heat Source	Voltage Correction Under-Relaxation 1						
Electrical Parameters							
Nominal Cell Capacity (ah) 14.6							
- Solution Options	C-Rate 1						
Specified C-Rate	System Current (a) 1						
<ul> <li>Specified System Current</li> <li>Specified System Voltage</li> </ul>	System Voltage (v) 4						
<ul> <li>Specified System Power</li> </ul>	System Power (w) 200						
Specified Resistance	External Resistance (ohm) 1						
Using Profile	Min. Stop Voltage (v) 3						
Set in Boundary Conditions							
Max. Stop Voltage (v) 4.3							
OK Init Reset Apply Cancel Help							

- b. Under the **Model Options** tab (Figure 25.2: Model Options (p. 1010)), configure the following battery operation conditions:
  - i. Under E-Chemistry Models, enable NTGK Empirical Model.
  - ii. In the Electrical Parameters group, retain the default value of 14.6 Ah for Nominal Cell Capacity.
  - iii. Enable Specified System Power in the Solution Options group and set System Power to 200
     W.
- c. Under the **Model Parameters** tab, retain the default settings for Y and U coefficients. For details, see the Fluent Advanced Add-On Modules documentation on the ANSYS Customer Portal (http://support.an-sys.com/documentation).
- d. Under the **Conductive Zones** tab (Figure 25.3: Conductive Zones (p. 1011)), configure the following settings:

Group	Control or List	Value or Selection
Active Components	Zone (s)	cell_1
		cell_2
		cell_3
Tab Components	Zone (s)	n_tabzone_1
		n_tabzone_2
		n_tabzone_3
		p_tabzone_1
		p_tabzone_2
		p_tabzone_3
Busbar Components	Zone (s)	bar1
		bar2

### Figure 25.3: Conductive Zones

SMD Battery Model								
Enable MSMD Battery Model								
Model Options Model Parameters	Conductive Zones	Electric Contact	s Advanced Option					
Active Components	Tab Components		Busbar Components					
Zone(s) [3/11]	Zone(s) [6/11]		one(s) [2/11]	x- <del>-</del>				
barl	barl		bar1	E				
bar2	bar2		bar2					
cell_1	cell_1		cell_1					
cell_2	cell_2		cell_2					
cell_3 n_tabzone_1	cell_3 n_tabzone_1		cell_3 n_tabzone_1					
n_tabzone_2	n_tabzone_2		n_tabzone_2					
n_tabzone_3	n_tabzone_3		n_tabzone_3					
p_tabzone_1	p_tabzone_1		p_tabzone_1					
p_tabzone_2	p_tabzone_2		p_tabzone_2					
p_tabzone_3	p_tabzone_3		p_tabzone_3					
				-				
۲ است ا								
OK Init Reset Apply Cancel Help								

e. Under the **Electric Contacts** tab (Figure 25.4: Electric Contacts (p. 1012)), configure the contact surface and external connector settings as follows:

Group	Control or List Value or Selection	
External Connectors	Negative Tap	tab_n
	Positive Tap	tab_p

The corresponding current or voltage boundary condition will be applied to those boundaries automatically.

MSMD Battery Model				<b>×</b>
Enable MSMD Battery Model				
Model Options Model Parameters	Conductive Zones	Electric Contacts	Advanced Option	
Contact Surfaces				]
Zone(s) [0/13]			Contact Resistance	e (ohm-m2)
tab_n tab_p wall-bar1 wall-cell_1 wall-cell_2 wall-cell_3 wall-n_tabzone_1 wall-n_tabzone_2 wall-n_tabzone_3				
Use virtual battery connection External Connectors Negative Tab [1/13]		itive Tab [1/13]		
tab_n         tab_p         wall-bar1         wall-cell_1         wall-cell_2         wall-cell_3         wall-n_tabzone_1         wall-n_tabzone_2         wall-n_tabzone_3         """"""""""""""""""""""""""""""""""""		b_n b_p all-bar1 all-bar2 all-cell_1 all-cell_2 all-cell_3 all-n_tabzone_1 all-n_tabzone_2 all-n_tabzone_3		
OK	Init Reset App	ly Cancel Help		

#### f. Click the Print Battery System Connection Information button.

ANSYS Fluent prints the battery connection information in the console window:

```
Battery Network Zone Information:

Battery Serial 1

Parallel 1

N-Tab zone: n_tabzone_1

Active zone: cell_1

P-Tab zone: p_tabzone_1

Battery Serial 2

Parallel 1

N-Tab zone: n_tabzone_2
```

```
Active zone: cell_2

P-Tab zone: p_tabzone_2

Battery Serial 3

Parallel 1

N-Tab zone: n_tabzone_3

Active zone: cell_3

P-Tab zone: p_tabzone_3

------

Busbar zone 1: barl

Busbar zone 2: bar2

Active zone list at odd level: cell_1 cell_3

Active zone list at even level: cell_2

Number of battery seriel =3; Number of parallel per serial=1

***************************
```

g. Verify that the connection information is correct. If an error message appears or if the connections are not what you want, redefine the conductive zones in the **Conductive Zones** tab (Figure 25.3: Conductive Zones (p. 1011)). Repeat this process until you confirm that the battery connections are set correctly.

#### Important

To set a valid connection, you must connect the negative tab to the positive tab through conductive zones.

h. Click OK to close the MSMD Battery Model dialog box.

In the background, Fluent automatically hooks all the necessary UDFs for the problem.

### 25.4.4.2. Defining New Materials

Define the new e\_material material for all the battery's cells and busbar\_material material for the battery pack's busbars and tabs.

#### Important

For the battery active material, you must define the electric conductivity via the UDS diffusivity on a per-scalar basis. ANSYS Fluent will use these two UDS scalars to solve the differential equations; for details, see the Fluent Advanced Add-On Modules documentation on the ANSYS Customer Portal (http://support.ansys.com/documentation).

For the battery passive material(s), you must define the UDS diffusivity using a user-defined function (UDF).

1. Create the electric material.



Create/Edit Materials		<b>—</b> ———————————————————————————————————		
Name	Material Type	Order Materials by		
e_material	solid	<ul> <li>Name</li> </ul>		
Chemical Formula	Fluent Solid Materials	Chemical Formula		
e	e_material (e)	Fluent Database		
	Mixture none	User-Defined Database		
Properties				
Cp (Specific Heat) (j/kg-k) constant	▼ Edit )			
871				
Thermal Conductivity (w/m-k) constant	Edit			
20	=			
UDS Diffusivity (kg/m-s) defined-per-	uds 🔹 Edit			
Electrical Conductivity (siemens/m) constant	▼ [Edit] -			
Change/Create Delete Close Help				

- a. In the **Create/Edit Materials** dialog box, enter **e\_material** for **Name** and **e** for **Chemical Formula**.
- b. Set Thermal Conductivity to 20.
- c. Under **Properties**, ensure that **define-per-uds** is selected from the **UDS Diffusivity** drop-down list and click **Edit...** next to **UDS Diffusivity**.
- d. In the **UDS Diffusion Coefficients** dialog box, set the constant value of 1.0 e6 for the both user-defined scalars.

💶 UDS Diffusion Coefficients 🧮	<
User-Defined Scalar Diffusion	
uds-0	
uds-1	
Coefficient (kg/m-s)	_
constant   Edit	
1000000	
OK Cancel Help	.#

- i. Select uds-0 in the User-Defined Scalar Diffusion list.
- ii. Retain **constant** from the **Coefficient** drop-down list.
- iii. Set **1.0** e6 [s/m] for **Coefficient**.

iv. In a similar way, set uds-1 to 1.0 e6 and close the UDS Diffusion Coefficients dialog box.

#### Note

- The units for UDS Diffusivity are s/m, and cannot be modified.
- In this tutorial, the electric conductivities are the same for the two scalars UDS0 and UDS1, so you will need to define only one material for all cell zones. If the electric conductivities are different for positive and negative electrodes, you will need to define two different materials, as described in the Fluent Advanced Add-On Modules documentation on the ANSYS Customer Portal (http://support.ansys.com/documentation).
- v. In the **Question** dialog box, click **No** to retain **aluminum** and add the new material (e\_material) to the materials list.
- e. Ensure that e\_material (e) is selected from the Fluent Solid Materials drop-down list.

#### f. Click Change/Create.

#### Note

If the UDS Diffusion Coefficients are defined through the defined-per-uds option, the Fluent solver does not use the value for **Electrical Conductivity**.

2. Create the busbar\_material material for busbars and tabs by modifying e-material you have created in the previous step.

<b>E</b> Setup $\rightarrow$ Materials $\rightarrow$ Solid $\rightarrow$ e-material	₫́ Edit
---	---------

Create/Edit Materials				<b>x</b>
Name		Material Type		Order Materials by
busbar_material		solid		Name
Chemical Formula		Fluent Solid Materials		Chemical Formula
bus		busbar_material (bus)	•	Fluent Database
		Mixture		User-Defined Database
		none	Ŧ	oser benned bacabasenn
Properties		1.1		
Cp (Specific Heat) (j/kg-k)	constant	- Edit ^		
	871			
Thermal Conductivity (w/m-k)	constant			
	20			
UDS Diffusivity (kg/m-s)	user-defined   Edit			
	battery_e_cond	l::msmdbatt		
Electrical Conductivity (siemens/m)	constant	▼ Edit		
	3.541e+07			
		-		
Change/Create Delete Close Help				

As stated in the problem description, you will use the same material for busbars and tabs.

#### Note

If the busbar and tab materials are different, you need to define the two different materials and assign them to the busbars and tabs, respectively.

The UDS Diffusivity for both busbar and tab materials must be defined through the UDF as described below.

- a. In the **Create/Edit Materials** dialog box, enter **busbar\_material** for **Name** and **bus** for **Chemical Formula**.
- b. Under Properties, select user-defined from the UDS Diffusivity drop-down list.
- c. In the User-Defined Functions dialog box that opens, verify that battery\_e\_cond::msmdbatt is selected and click OK.

User-Defined Functions	×
	_
battery_e_cond::msmdbatt	
	E
	-
OK Cancel Help	
	H.

d. In the **Question** dialog box, click **No** to retain **e-material** and add the new busbar\_material material to the materials list.

Fluent hooks the battery\_e\_cond::msmdbatt UDF to the busbar material.

- e. Ensure that busbar\_material (bus) is selected from the Fluent Solid Materials drop-down list.
- f. Set Thermal Conductivity to 20.
- g. Retain the default value of 3.541 e7 [1/ohm-m] for Electrical Conductivity.
- h. Click Change/Create.
- i. Close the Create/Edit Materials dialog box.

## 25.4.4.3. Defining Cell Zone Conditions

Assign e\_material to all the cell zones and busbar\_material to all the tabs and busbars.

1. Assign e\_material to the cell\_1 cell zone.

# **Setup** $\rightarrow$ **Cell** Zone Conditions $\rightarrow \stackrel{\frown}{=} cell_1 \rightarrow Edit...$

- a. In the **Solid** dialog box, select e\_material from the **Material Name** drop-down list.
- b. Click **OK**.
- 2. Copy the cell zone condition for the cell\_1 zone to the cell\_2 and cell\_3 cell zones.

**The Setup**  $\rightarrow$  **Cell Zone Conditions**  $\rightarrow$  **E cell\_1**  $\rightarrow$  **Copy...** 

- a. In the **Copy Conditions** dialog box, select cell\_1 in the **From Cell Zone** list.
- b. In the **To Cell Zones** list, select cell\_2 and cell\_3.
- c. Click Copy.
- d. Click **OK** in the **Question** dialog box to copy the cell zone conditions and close the **Copy Conditions** dialog box.
- 3. In a similar manner, assign busbar\_material to all the tabs and busbars cell zones.

## 25.4.4.4. Defining Boundary Conditions

Define the thermal boundary conditions for all walls for the cells, busbars, and tabs. The boundary conditions for the two UDSs have been set automatically when you defined the cell zone conditions.

1. Set the convection boundary condition for wall-cell\_1.

Setting Up Physics  $\rightarrow$  Zones  $\rightarrow$  Boundaries

- a. In the **Boundary Conditions** task page, double-click **wall-cell\_1**.
- b. In the Wall dialog box, under the Thermal tab, configure the following settings:
  - i. Under Thermal Conditions, enable Convention.
  - ii. Set Heat Transfer Coefficient to 5  $[w/m^2 K]$ .
  - iii. Set Free Stream Temperature to 300 [K].
  - iv. Click **OK** to close the **Wall** dialog box.
- 2. Copy the boundary conditions for wall-cell\_1 to wall-cell\_2, wall-cell\_3 and all the tab and busbar wall zones (a boundary zones that have names starting with the "wall" string and containing the "bar" or "tabzone" string).

**E** Setup  $\rightarrow$  **\bigcirc** Boudnary Conditions  $\rightarrow$  **\equiv** wall-cell  $1 \rightarrow$  Copy...

Copy Conditions				<b>—</b> ×
From Boundary Zone Filter Text	-	To Boundary Zones	Filter Text	x
tab_n	*	tab_n		
tab_p		tab_p		
wall-bar1		wall-bar1		
wall-bar2		wall-bar2		
wall-cell_1		wall-cell_2		
wall-cell_2		wall-cell_3		
wall-cell_3		wall-n_tabzone_1		
wall-n_tabzone_1		wall-n_tabzone_2		
wall-n_tabzone_2		wall-n_tabzone_3		
wall-n_tabzone_3	E	wall-p_tabzone_1		
wall-p_tabzone_1		wall-p_tabzone_2		
wall-p_tabzone_2	-	wall-p_tabzone_3		
	Сору	Close Help		

## 25.4.4.5. Specifying Solution Settings

1. Turn off the flow equation.

**Solving**  $\rightarrow$  Controls  $\rightarrow$  Equations...

- a. In the Equations dialog box, deselect Flow from the Equation selection list.
- b. Click OK.
- 2. Remove the convergence criteria to ensure that automatic convergence checking does not occur.

**Solving**  $\rightarrow$  Reports  $\rightarrow$  Residuals...

- a. In the Residual Monitors dialog box, select none from the Convergence Criterion drop-down list.
- b. Click **OK**.
- 3. Create a surface report definition for the voltage at the positive tab.

**Solving**  $\rightarrow$  Reports  $\rightarrow$  Definitions  $\rightarrow$  New  $\rightarrow$  Surface Report  $\rightarrow$  Area-Weighted Average

Surface Report Definition	
Name	Report Type
surf-mon-1	Area-Weighted Average
Options	Custom Vectors
	Vectors of
Per Surface	current-density-jn 🔻
Average Over	Custom Vectors
1	
	Field Variable
Report Files [0/0]	User Defined Scalars
	Potential Phi+
	Surfaces Filter Text
	interior-p_tabzone_2
	interior-p_tabzone_3
Report Plots [0/0]	V TX 4 Wall
	tab_n
	tab_p wall-bar1
	wall-bar2
	wall-cell_1
Create	wall-cell_2
Report File	wall-cell_3
Report Plot	wall-n_tabzone_1
Frequency 1	wall-n_tabzone_2 wall-n_tabzone_3
Print to Console	
Print to Console	Highlight Surfaces
Create Output Parameter	New Surface
	Compute Cancel Help

- a. In the **Surface Report Definition** dialog box, enter **surf-mon-1** for **Name**.
- b. Select User Defined Scalars... and Potential Phi+ from the Field Variable drop-down lists.
- c. From the **Surfaces** selection list, select tab\_p.
- d. In the Create group box, enable Report Plot and Print to Console.
- e. Click **OK** to save the surface report definition and close the **Surface Report Definition** dialog box.
- f. Modify the attributes of the plot axes.

```
Example 1 Solution \rightarrow Monitors \rightarrow Report Plots \rightarrow surf-mon-1-rplot \stackrel{\frown}{\rightarrow} Edit...
```

- i. In the **Edit Report Plot** dialog box, under the **Plot Window** group box, click the **Axes...** button to open the **Axes** dialog box.
- ii. Select the X axis and set Precision to 0.

- iii. Click Apply.
- iv. Select the Y axis and set Precision to 2.
- v. Set **Precision** to 2.
- vi. Click **Apply** and close the **Axes** dialog box.

#### Note

You must click **Apply** to save the modified settings for each axis.

vii. Click OK to close the Edit Report Plot dialog box.

- g. Ensure that time-step is selected from the Get Data Every drop-down list.
- h. Click **OK**.
- 4. Create a volume report definition to monitor the maximum temperature in the domain.

**Solving**  $\rightarrow$  Reports  $\rightarrow$  Definitions  $\rightarrow$  New  $\rightarrow$  Volume Report  $\rightarrow$  Max...

Volume Report Definition				
Name	Report Type			
vol-mon-1	Max			
Options	Field Variable			
	Temperature 🔻			
Per Zone	Static Temperature 🗸 🗸			
Average Over				
1	Cell Zones Filter Text			
	bar1			
Report Files [0/0]	bar2			
	cell_1 cell_2			
	cell_2 cell_3			
	n_tabzone_1			
	n_tabzone_2			
Report Plots [0/0]	n_tabzone_3			
	p_tabzone_1 p_tabzone_2			
	p_tabzone_2 p_tabzone_3			
Create				
🔲 Report File				
Report Plot				
Frequency 1				
Print to Console				
Create Output Parameter				
OK Compute Cancel Help				

- a. In the **Volume Report Definition** dialog box, enter **vol-mon-1** for **Name**.
- b. Select Temperature... and Static Temperature from the Field Variable drop-down lists.
- c. From the **Cell Zones** selection list, select all zones.
- d. In the Create group box, enable Report Plot and Print to Console.
- e. Click **OK** to save the volume report definition settings and close the **Volume Report Definition** dialog box.
- f. Modify the attributes of the plot axes.

```
Solution \rightarrow Monitors \rightarrow Report Plots \rightarrow vol-mon-1-rplot \stackrel{\frown}{\rightarrow} Edit...
```

i. In the **Edit Report Plot** dialog box, under the **Plot Window** group box, click the **Axes...** button to open the **Axes** dialog box.

- ii. Select the **X** axis and set **Precision** to **0**.
- iii. Click Apply.
- iv. Select the Y axis and set Precision to 2.
- v. Click **Apply** and close the **Axes** dialog box.
- vi. Ensure that time-step is selected from the Get Data Every drop-down list.
- vii. Click OK to close the Edit Report Plot dialog box.
- 5. Save the case file.

**File**  $\rightarrow$  Write  $\rightarrow$  Case...

## 25.4.4.6. Obtaining Solution

1. Initialize the field variables using the **Standard Initialization** method.

### **F**Solution $\rightarrow$ Solution Initialization

- a. Retain the selection of **Standard** from the **Initialization Methods** group box.
- b. Click Initialize.

You do not need to modify the **Initial Values** in the **Solution Initialization** task page, because these values are not used for initialization. The ANSYS Fluent solver automatically computes the initial condition for UDS0 and UDS1.

2. Run the simulation.

## Solving $\rightarrow$ Run Calculation

- a. Set Time Step Size to 30 seconds and No. of Time Steps to 50.
- b. Click **Calculate** and run the simulation up to 1500 seconds.

The residual plot, the history of the voltage at the positive tap and the history of the maximum temperature in the domain are shown in Figure 25.5: Residual History of the Simulation (p. 1023), Figure 25.6: Surface Report Plot of Discharge Curve at 200W (p. 1023), and Figure 25.7: Volume Report Plot of Maximum Temperature in the Domain (p. 1024), respectively.

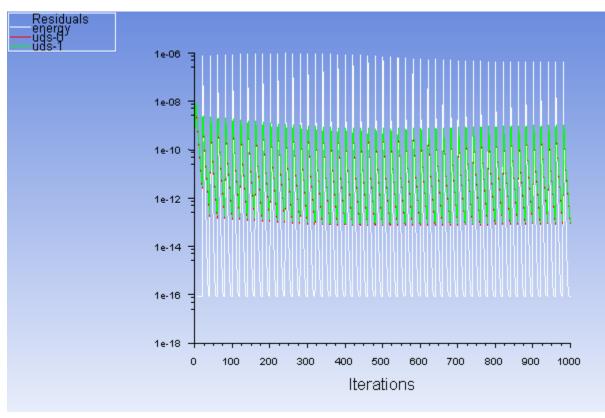
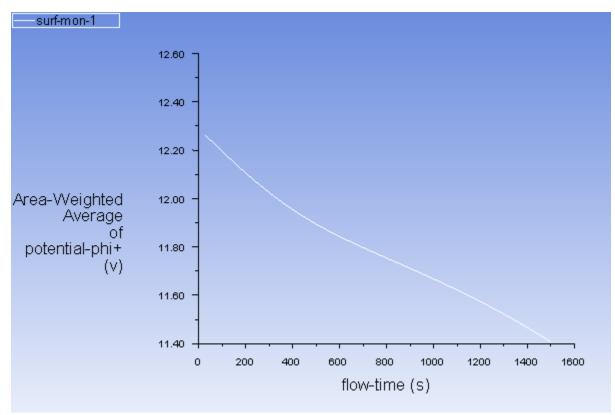


Figure 25.5: Residual History of the Simulation





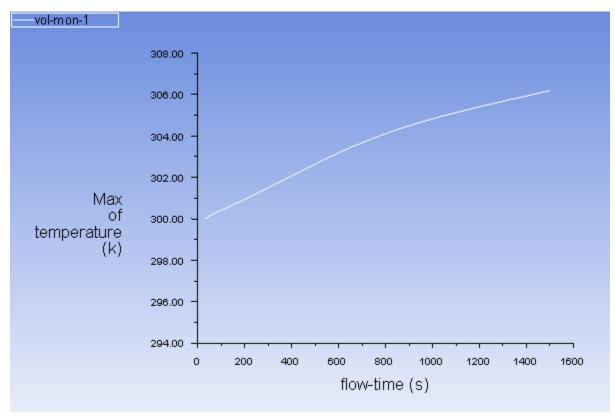


Figure 25.7: Volume Report Plot of Maximum Temperature in the Domain

## 25.4.5. Postprocessing

In this section, postprocessing options for the MSMD battery model solution are presented.

1. Display the vector plot of current density.

<b>2</b> Vectors			×	
Options	Vectors of			
Global Range	current-density-j			
Auto Range	Color by			
Clip to Range	User Defined Memory			
<ul> <li>Auto Scale</li> <li>Draw Mesh</li> </ul>	Magnitude of Current Density 🗸			
	Min (a/m2)	Max (a/m2)		
Style	318.5375	749153.7		
arrow 🔻	Surfaces Filter Text	=	ō F. 🗾 🗔	
0.003 0 🚖	Internal		•	
Vector Options Custom Vectors	✓ Wall tab_n tab_p wall-bar1 wall-bar2		T	
	New Surface			
	Display Compute	Close Help	H	

- a. In the Vectors dialog box, select current-density-j from the Vectors of drop-down list.
- b. Select **User Defined Memory...** and **Magnitude of Current Density** from the **Color by** drop-down list.
- c. Click the **Toggle Tree View** button next to the **Surfaces** filter and from the drop-down list, select **Surface Type** (under **Group By**).
- d. From the **Surfaces** selection list, select **Wall**.

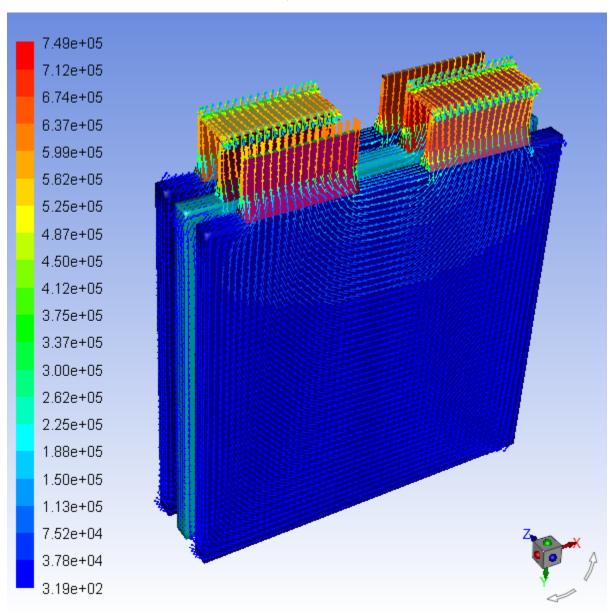
The surfaces of the "wall" type are automatically selected in the **Surfaces** list.

- e. In the **Options** group, enable **Draw Mesh** and set the mesh display options as desired.
- f. Set Scale to 0.003.
- g. Click Vector Options....
  - i. In the Vector Options dialog box, enable Fixed Length.

Vector Options	×	
In Plane Fixed Length X Component Y Component Z Component	Scale Head 0.1 Color	
Apply Close Help		

All vectors in your plot will be displayed with the same lengths.

- ii. Click **Apply** and close the **Vector Options** dialog box.
- h. Click **Display** and close the **Contours** dialog box.



**Figure 25.8: Vector Plot of Current Density** 

#### Note

Use the **Headlight** and **Lighting** display options under the **Viewing** ribbon tab to manipulate the graphics display.

2. Display the contour plot of the temperature.

Contours		×
Options <ul> <li>Filled</li> <li>Node Values</li> <li>Global Range</li> <li>Auto Range</li> <li>Clip to Range</li> </ul>	Contours of Temperature	•
	Static Temperature	•
	Min (k) Max (k) 305.5614 306.1825	
Draw Profiles	Surfaces Filter Text	) <b>-</b>
<ul> <li>Draw Mesh</li> <li>Coloring</li> <li>Banded</li> <li>Smooth</li> <li>Levels Setup</li> <li>20 1 +</li> </ul>	Internal Wall tab_n tab_p wall-bar1 wall-bar2 wall-cell_1 New Surface  Display Compute Close Help	• III

- a. In the **Contours** dialog box, in the **Options** group box, enable **Filled**.
- b. From the **Contours of** drop-down list, select **Temperature...** and **Static Temperature**.
- c. Click the **Toggle Tree View** button next to the **Surfaces** filter and from the drop-down list, select **Surface Type** (under **Group By**).
- d. From the **Surfaces** selection list, select **Wall**.
- e. Click **Display** and close the **Contours** dialog box.

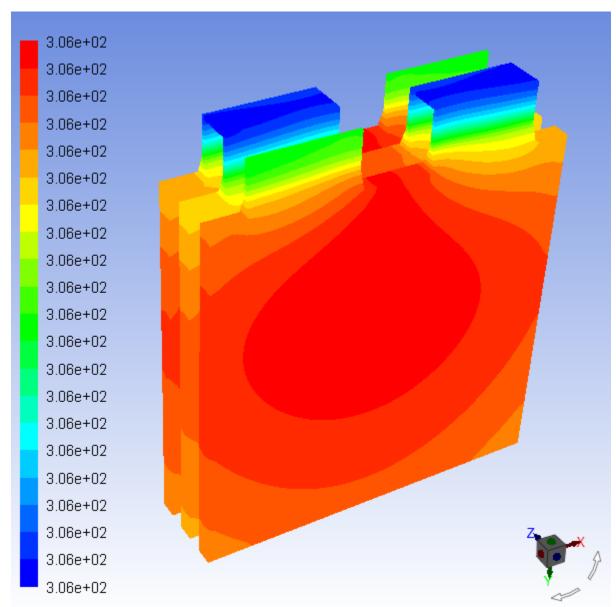


Figure 25.9: Contour Plot of Temperature

3. In a similar manner, display the contour of Ohmic heat source.

Contours	
Options <ul> <li>Filled</li> <li>Node Values</li> <li>Global Range</li> <li>Auto Range</li> <li>Clip to Range</li> </ul>	Contours of User Defined Memory
	Volumetric Ohmic Source            Min (w/m3)         Max (w/m3)           0.2764559         35035.46
Draw Profiles	Surfaces Filter Text
Coloring Banded Smooth Levels Setup	Wall tab_n tab_p wall-bar1 wall-bar2 wall cell 1
20 🔹 1 🔹	wall-cell_1  New Surface   Display Compute Close Help

- a. From the **Contours of** drop-down list, select **User Defined Memory...** and **Volumetric Ohmic Source**.
- b. Click **Display**.

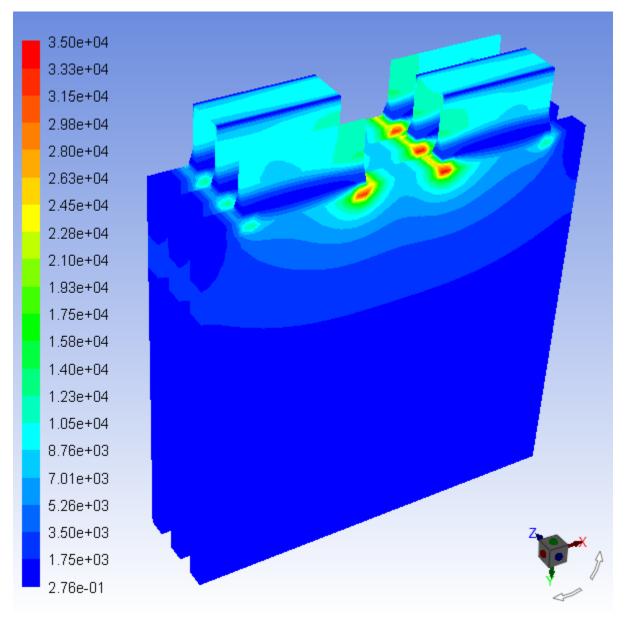
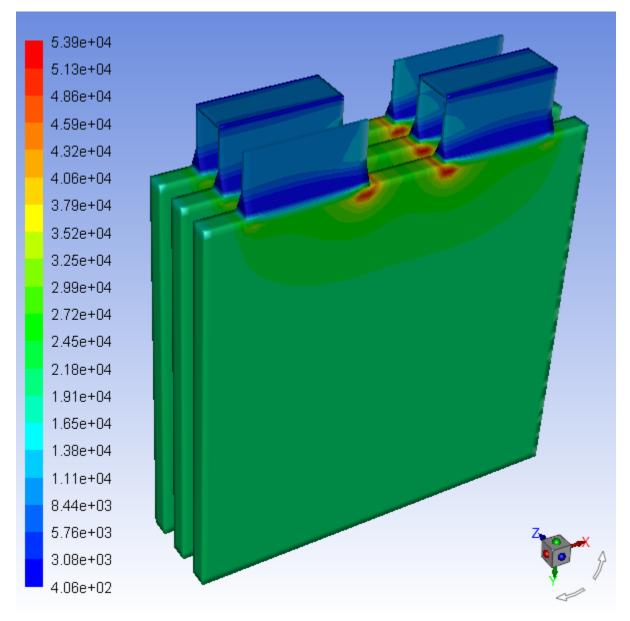


Figure 25.10: Ohmic Heat Generation Rate

4. In a similar manner, display the contour of the total heat source.

Contours	
Options Filled Global Range Auto Range	Contours of User Defined Memory
	Total Heat Generation Source                Min (w/m3)             Max (w/m3)               405.7072             53946.82
Clip to Range Draw Profiles Draw Mesh	Surfaces Filter Text
Coloring Banded Smooth Levels Setup 20 1	V Internal ▲ Wall tab_n tab_p wall-bar1 wall-bar2 wall-cell_1 New Surface ▼
	Display Compute Close Help

- a. From the **Contours of** drop-down list, select **User Defined Memory...** and **Total Heat Generation Source**.
- b. Click **Display**.



### Figure 25.11: Total Heat Generation Rate

# 25.5. Summary

This tutorial has demonstrated the use of the MSMD battery add-on to perform electrochemical and heat transfer simulations for battery packs. You have learned how to set up and solve the problem for the battery pack of the 1P3S configuration using the NTGK Battery submodel. You have also learned some of the postprocessing capabilities available in the MSMD battery model.

As an exercise, you can obtain solutions using the Equivalent Circuit Model and Newman P2D Model. For details, see the Fluent Advanced Add-On Modules documentation on the ANSYS Customer Portal (http://support.ansys.com/documentation).